

IPC-2222

Sectional Design Standard for Rigid Organic Printed Boards

IPC-2222
February 1998

Supersedes IPC-D-275
September 1991

A standard developed by IPC

2215 Sanders Road, Northbrook, IL 60062-6135
Tel. 847.509.9700 Fax 847.509.9798
www.ipc.org

The Principles of Standardization

In May 1995 the IPC's Technical Activities Executive Committee adopted Principles of Standardization as a guiding principle of IPC's standardization efforts.

Standards Should:
- Show relationship to Design for Manufacturability (DFM) and Design for the Environment (DFE)
- Minimize time to market
- Contain simple (simplified) language
- Just include spec information
- Focus on end product performance
- Include a feedback system on use and problems for future improvement

Standards Should Not:
- Inhibit innovation
- Increase time-to-market
- Keep people out
- Increase cycle time
- Tell you how to make something
- Contain anything that cannot be defended with data

Notice

IPC Standards and Publications are designed to serve the public interest through eliminating misunderstandings between manufacturers and purchasers, facilitating interchangeability and improvement of products, and assisting the purchaser in selecting and obtaining with minimum delay the proper product for his particular need. Existence of such Standards and Publications shall not in any respect preclude any member or nonmember of IPC from manufacturing or selling products not conforming to such Standards and Publication, nor shall the existence of such Standards and Publications preclude their voluntary use by those other than IPC members, whether the standard is to be used either domestically or internationally.

Recommended Standards and Publications are adopted by IPC without regard to whether their adoption may involve patents on articles, materials, or processes. By such action, IPC does not assume any liability to any patent owner, nor do they assume any obligation whatever to parties adopting the Recommended Standard or Publication. Users are also wholly responsible for protecting themselves against all claims of liabilities for patent infringement.

IPC Position Statement on Specification Revision Change

It is the position of IPC's Technical Activities Executive Committee (TAEC) that the use and implementation of IPC publications is voluntary and is part of a relationship entered into by customer and supplier. When an IPC standard/guideline is updated and a new revision is published, it is the opinion of the TAEC that the use of the new revision as part of an existing relationship is not automatic unless required by the contract. The TAEC recommends the use of the lastest revision. Adopted October 6. 1998

Why is there a charge for this standard?

Your purchase of this document contributes to the ongoing development of new and updated industry standards. Standards allow manufacturers, customers, and suppliers to understand one another better. Standards allow manufacturers greater efficiencies when they can set up their processes to meet industry standards, allowing them to offer their customers lower costs.

IPC spends hundreds of thousands of dollars annually to support IPC's volunteers in the standards development process. There are many rounds of drafts sent out for review and the committees spend hundreds of hours in review and development. IPC's staff attends and participates in committee activities, typesets and circulates document drafts, and follows all necessary procedures to qualify for ANSI approval.

IPC's membership dues have been kept low in order to allow as many companies as possible to participate. Therefore, the standards revenue is necessary to complement dues revenue. The price schedule offers a 50% discount to IPC members. If your company buys IPC standards, why not take advantage of this and the many other benefits of IPC membership as well? For more information on membership in IPC, please visit www.ipc.org or call 847/790-5372.

Thank you for your continued support.



IPC-2222

Sectional Design Standard for Rigid Organic Printed Boards

Developed by the IPC-D-275 Task Group (D-31b) of the Rigid Printed Board Committee (D-30) of IPC

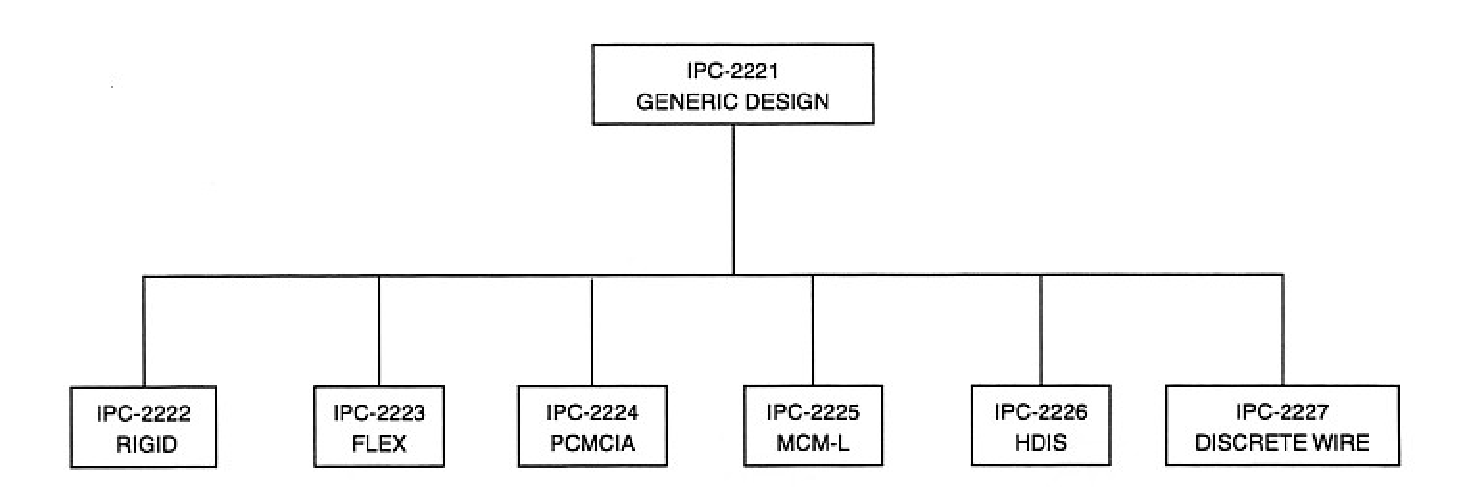


Users of this standard are encouraged to participate in the development of future revisions.

Contact:

IPC
2215 Sanders Road
Northbrook, Illinois
60062-6135
Tel 847 509.9700
Fax 847 509.9798

HIERARCHY OF IPC DESIGN SPECIFICATIONS
(2220 SERIES)



FOREWORD

This standard is intended to provide information on the detailed requirements for organic rigid printed board design. All aspects and details of the design requirements are addressed to the extent that they can be applied to the unique requirements of those designs that use organic rigid (reinforced) materials or organic materials in combination with inorganic materials (metal, glass, ceramic, etc.) to provide the structure for mounting and interconnecting electronic, electromechanical, and mechanical components.

The information contained herein is intended to supplement generic engineering considerations and design requirements identified in IPC-2221. When coupled with the engineering design input, the complete disclosure should facilitate the appropriate selection process of the materials and the detailed organic rigid structure fabrication technology necessary to meet the engineering design objectives.

The selected component mounting and interconnecting technology for the printed board should be commensurate with the requirements provided and the specific focus of this sectional document.

IPC's documentation strategy is to provide distinct documents that focus on specific aspect of electronic packaging issues. In this regard document sets are used to provide the total information related to a particular electronic packaging topic. A document set is identified by a four digit number that ends in zero (0).

Included in the set is the generic information which is contained in the first document of the set and identified by the four digit set number. The generic standard is supplemented by one or many sectional documents each of which provide specific focus on one aspect of the topic or the technology selected. The designer of the printed board, needs as a minimum, the generic, the sectional of the chosen technology, the generic engineering considerations, and the engineering description of the final product.

Failure to have all information available prior to starting a design may result in a product that is difficult to manufacture or exceeds the cost predictions or expectations of the printed board.

As technology changes, specific focus standards will be updated, or new focus standards added to the document set. The IPC invites input on the effectiveness of the documentation and encourages user response through completion of "Suggestions for Improvement" forms located at the end of each document

Acknowledgment

Any Standard involving a complex technology draws material from a vast number of sources. While the principal members of the IPC-D-275 Task Group (D-31b) of the Rigid Printed Board Committee (D-30) are shown below, it is not possible to include all of those who assisted in the evolution of this Standard. To each of them, the members of the IPC extend their gratitude.

Rigid Printed Board Committee

Chairman
Bob Neves
Microtek Lab

IPC-D-275 Task Group (D-31b)

Chairman
Lionel Fullwood
Wong's Kong King Int'l

Technical Liaison of the IPC Board of Directors

Ronald Underwood
Circuit Center

IPC-D-275 Task Group

Richard Altenhofen, Motorola GSTG
Daniel Arnold, EMD Associates Inc.
Lance A. Auer, Hughes Missile Systems Company
Nanci J. Baggett, Printed Circuit Resources
Steve Bakke, Alliant Techsystems Inc.
Karl J. Bates, Lucent Technologies
Robert E. Beauchamp, Lockheed Martin Missiles & Space
Frank Belisle, Sundstrand Aerospace
David W. Bittle, Raytheon Aircraft Company
Daniel L. Botts, Hughes Training, Inc.
John Bourque, Shure Brothers Inc.
Scott A. Bowles, Sovereign Circuits Inc.
Stephen G. Bradley, CAL Corporation
Jim Brock, SCI Systems Inc.
Ignatius Chong, Celestica
David J. Corbett, DSCC
Brian Crowley, Hewlett Packard Laboratories
Georgia DeGrandis, ABB Ceag Power Supplies Inc.
Yong Deng, Owens-Corning Fiberglass Corp.
Michele J. DiFranza, The Mitre Corp.
C. Don. Dupriest, Lockheed Martin Vought Systems
Theodore Edwards, Honeywell Inc.
Will J. Edwards, Lucent Technologies Inc.
Werner Engelmaier, Engelmaier Associates, Inc.
Thomas R. Etheridge, McDonnell Douglas Aerospace
Joe Fjelstad, Tessera Inc.
Martin G. Freedman, Amp Inc.
Lionel Fullwood, Wong's Kong King Int'l
Mahendra S. Gandhi, Hughes Aircraft Co.
Paul Grande, Jr., U.S. Navy
Michael R. Green, Lockheed Martin Missiles & Space
Lyle F. Harford, Texas Instruments Inc.
Andrew J. Heidelberg, Micron Custom Mfg. Services Inc.
Ralph J. Hersey, Ralph Hersey & Associates
Phillip E. Hinton, Hinton -PWB-Engineering
Octavian Iordache, Circo Craft Co. Inc.
Don Jensen, Endicott Research Group
Arturo J. Jordan, Pollak Trnsprtatn Electrnics Div
John A. Kelly, Motorola GSTG
Therese Kokocinski, Northrop Grumman Corporation
Stephen Korchynsky, Lockheed Martin Federal Systems
George T. Kotecki, Northrop Grumman Corporation
Thomas E. Kurtz, Hughes Defense Communications
Clifford H. Lamson, Harris Corp.
Bonnie Lauch, Honeywell Inc.
Stan C. Mackzum, Ericsson Inc.
James F. Maguire, Boeing Defense & Space Group
David J. Malanchuk, Eastman Kodak Co. KAD
Wesley R. Malewicz, Siemens Medical Systems Inc.
Susan Mansilla, Robisan Laboratory Inc.
Lester Mielczarek, CAE Electronics Ltd.
Kelly J. Miller, CAE Electronics Ltd.
John H. Morton, Lockheed Martin Federal Systems
Karl B. Mueller, Hughes Aircraft Co.
Joseph L. Mulcahy, Methode Electronics Inc. East
Benny Nilsson, Ericsson Telecom AB
R. Bruce. Officer, Sanders, A Lockheed Martin Co.
Scott S. Opperhauser, Trace Laboratories - East
John Papinko, Gulton Data Systems
Ron Payne, Primex Aerospace
Richard Peyton, Lockheed Martin Astronautics
Larry L. Puckett, Sandia National Labs Albuquerque
Paul J. Quinn, Lockheed Martin Missiles & Space
Kurt Ravenfeld, Lockheed Martin Corporation
Randy R. Reed, Merix Corporation
Bruce C. Rietdorf, Hughes Defense Communications
Jerald G. Rosser, Hughes Missile Systems Company
Vincent J. Ruggeri, Raytheon Company
Don W. Rumps, Lucent Technologies Inc.
Robert Russell, Texas Instruments Inc.
Merlyn L. Seltzer, Hughes Delco Systems Operations
Nusrat Sherali, IBM Corp.
Lowell Sherman, DSCC

Rae Shyne, Prototron Circuits Inc.

Grant (Rick) W. Smedley, III, Printed Circuit Resources

E. Lon. Smith, Lucent Technologies Inc.

Joseph J. Sniezek, IBM Corp./ Endicott Electronic Pa

William F. Spurny, AlliedSignal Aerospace

Robert J. St. Pierre, New England Laminates

Thomas K. Stewart, Speedy Circuits

Gil Theroux, Honeywell Inc.

Ronald E. Thompson, U.S. Navy

Max E. Thorson, Compaq Computer Corporation

Lutz E. Treutler, Fachverband Elektronik Design

Robert Vanech, Northrop Grumman Norden Systems

Eric L. Vollmar, Methode Electronics Inc.

Forrest L. Voss, Rockwell International

Rich Warzecha, Advanced Flex Inc.

Clark F. Webster, Computing Devices International

David A. White, Input/Output Inc.

Table of Contents

Figures

Tables

Sectional Design Standard for Rigid Organic Printed Boards

1.0 SCOPE

This standard establishes the specific requirements for the design of rigid organic printed boards and other forms of component mounting and interconnecting structures. The organic materials may be homogeneous, reinforced, or used in combination with inorganic materials; the interconnections may be single, double, or multilayered.

1.1 Purpose The requirements contained herein are intended to establish specific design details that **shall** be used in conjunction with IPC-2221 (see 2.0) to produce detailed designs intended to mount and attach passive and active components.

The components may be through-hole, surface mount, fine pitch, ultra-fine pitch, array mounting or unpackaged bare die. The materials may be any combination able to perform the physical, thermal, environmental, and electronic function.

1.2 Document Hierarchy Document hierarchy **shall** be in accordance with the generic standard IPC-2221.

1.3 Presentation Presentation **shall** be in accordance with the generic standard IPC-2221.

1.4 Interpretation Interpretation **shall** be in accordance with the generic standard IPC-2221.

1.5 Classification of Products Classification of Products **shall** be in accordance with the generic standard IPC-2221 and as follows:

1.5.1 Board Type This standard provides design information for different board types. Board types are classified as:

Type 1 — Single-Sided Printed Board
Type 2 — Double-Sided Printed Board
Type 3 — Multilayer Board without Blind or Buried Vias
Type 4 — Multilayer Board with Blind and/or Buried Vias
Type 5 — Multilayer Metal-Core Board without Blind or Buried Vias
Type 6 — Multilayer Metal-Core Board with Blind and/or Buried Vias

1.6 Assembly Types A type designation signifies further sophistication describing whether components are mounted on one or both sides of the packaging and interconnecting structure. Type 1 defines an assembly that has components mounted on only one side; Type 2 is an assembly with components on both sides. Type 2, Class A is not recommended.

Figure 1-1 shows the relationship of two types of assemblies.

The need to apply certain design concepts should depend on the complexity and precision required to produce a particular land pattern or P&I structure. Any design class may be applied to any of the end-product equipment categories; therefore, a moderate complexity (Type 1B) would define components mounted on one side (all surface mounted) and when used in a Class 2 product (dedicated service electronics) is referred to as Type 1B, Class 2. The product described as a Type 1B, Class 2 might be used in any of the end-use applications; the selection of class being dependent on the requirements of the customers using the application.

2.0 APPLICABLE DOCUMENTS

The following documents form a part of this document to the extent specified herein. If a conflict of requirements exist between IPC-2222 and those listed below, IPC-2222 takes precedence.

The revision of the document in effect at the time of solicitation **shall** take precedence.

2.1 Institute for Interconnecting and Packaging Electronic Circuits (IPC)[1]

IPC-EG-140 Specification For Finished Fabric Woven From "E" Glass for Printed Board

IPC-MF-150 Metal Foil for Printed Wiring Applications

IPC-CF-152 Composite Metallic Materials Specification for Printed Wiring Boards

IPC-D-279 Design Guidelines for Reliable Surface Mount Technology Printed Board Assemblies

IPC-TM-650 Test Methods Manual[2]

Method 2.1.1 Microsectioning

Method 2.1.6 Thickness of Glass Fabric

IPC-SM-782 Surface Mount Design and Land Pattern Standard

1. IPC, 2215 Sanders Road, Northbrook, IL 60062
2. Current and revised IPC Test Methods are available through IPC-TM-650 subscription and on the IPC Web site (www.ipc.org/html/testmethods.htm).

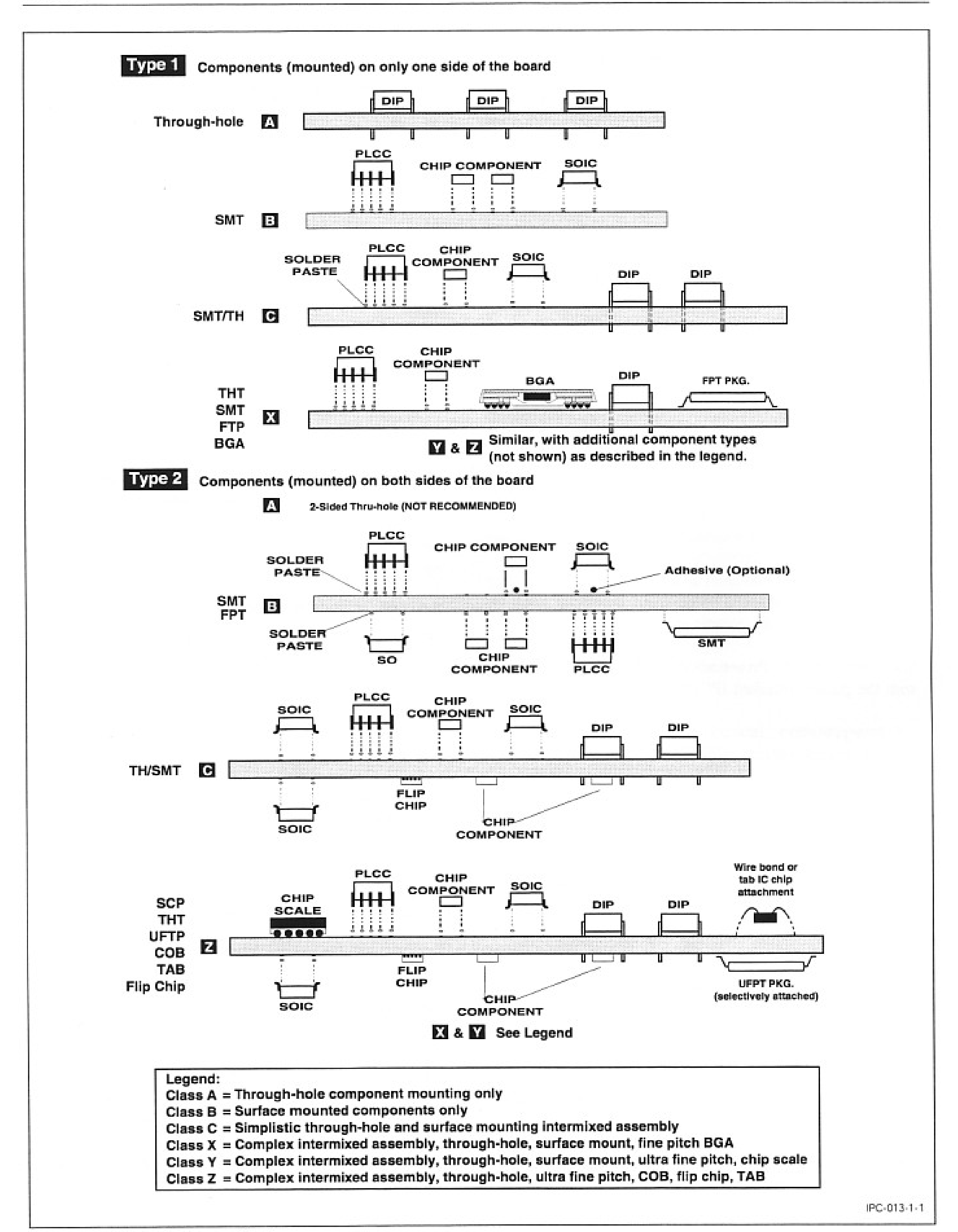


Figure 1-1 Electrical assembly types

IPC-2221 Generic Standard on Printed Board Design

IPC-4101 Laminate/Prepreg Materials Standard for Printed Boards

IPC-6012 Qualification and Performance Specification for Rigid Printed Boards

2.2 Underwriters Laboratories[3]

UL 746E Standard Polymeric Materials, Materials Used in Printed Wiring Boards

3.0 GENERAL REQUIREMENTS

General requirements **shall** be in accordance with the generic standards IPC-2221.

3.1 Performance Requirements Finished rigid printed boards shall meet the performance requirements of IPC-6012.

4.0 MATERIALS

4.1 Material Selection Material Selection **shall** be in accordance with the generic standard IPC-2221.

4.2 Dielectric Base Materials (Including Prepregs and Adhesives) Dielectric base materials **shall** be in accordance with the generic standard IPC-2221 and the following:

4.2.1 Epoxy Laminates Epoxies are the most common resin materials which are combined with glass cloth to produce laminates. When compared to other laminate materials, epoxies offer advantages in availability and relative ease in processing. The many different types and blends of epoxies exhibit a wide range of selection for usage or soldering processes; epoxies with a T_g (glass transition temperature) from 110 to 120°C up to 180 to 190°C are available from most laminate suppliers with some of the most used in the 135 to 145°C range.

4.2.2 High-Temperature Laminates High temperature laminates include those made from resins such as Epoxy, Cyanate Ester, Triazine blends and polyimide. High temperature resin laminates offer the advantages of increased chemical and temperature resistance. Disadvantages include the need for specialized processing and higher material cost.

4.2.3 Special Clad Materials The use of surface mount technology may require the use of special clad materials when coefficient of thermal expansion matching is critical. Examples of these special materials are copper-clad Invar, epoxy or polyimide with aramid fiber and polyimide/quartz. The most common usage is for Class 3 boards, although there may also be some application for Class 2. These materials offer the advantages of performance for specialized applications; the need for unique processing during board fabrication is a disadvantage.

4.2.4 Other Laminates Laminates, such as paper-based phenolics etc., have acceptance in some consumer products where the complexity is quite low due to lesser material and manufacturing costs. These materials are associated with very high volume products with lower performance requirements than those usually associated with epoxy type laminates.

4.3 Laminate Materials Laminate materials **shall** be in accordance with the generic standard IPC-2221 and as follows:

When metal clad, foil type **shall** be as specified in IPC-MF-150. Unclad laminates without an adhesive, per IPC-4101 may be used as fillers in multilayer boards for dielectric spacing between layers.

When Underwriter's Labs (UL) requirements are imposed, the material used must be approved by UL for use as fillers in multilayer boards for dielectric spacing between layers. Printed boards **shall** be fabricated from the laminate materials specified in Table 4-1 or UL 746E.

The board design **shall** be such that internal temperature rise due to current flow in the conductor, when added to all other sources of heat at the conductor/laminate interface, will not result in an operating temperature in excess of that specified for the laminate material. The values in Table 4-1 are based on long term thermal aging tests by UL and may be mandatory for designs to be used in UL approved products. Since heat dissipated by parts mounted on the boards will contribute local heating effects, the material selection **shall** take this factor, plus the equipment's general internal rise temperature, plus the specified operating ambient temperature for the equipment into account. Hot spot temperatures **shall** not exceed the temperatures specified in Table 4-1 for the laminate material selected. Materials used (copper-clad, prepreg, copper foil, heat sink, etc.) **shall** be specified on the master drawing.

4.3.1 Measurement of Dielectric Thickness Dielectric thickness will vary across applications. Thickness by mechanical measurement is determined in accordance with IPC-TM-650, Method 2.1.6. Thickness by microsection (view shown in Figure 4-1) is determined in accordance with IPC-TM-650, Method 2.1.1. The dielectric thickness is measured in accordance with Figure 4-1 and taken at the closest point between metal claddings.

3. Underwriter's Labs, 333 Pfingsten Avenue, Northbrook, IL 60062

Table 4-1 Clad Laminate Maximum Operating Temperatures[1]

Designation		Dielectric Thickness (min)	Temperature[4, 5, 6, 7] (max)
NEMA	IPC-4101		
FR-4	21/24/25/27	0.1 mm	120°C
		0.4 mm	130°C
FR-5	23	0.6 mm	140°C
		1.4 mm	170°C
GPY	40/41/42	0.1 mm	140°C
		1.6 mm	170°C
	50/52	0.1 mm	120°C
		0.4 mm	130°C
	51/53/60	0.1 mm	140°C
		1.6 mm	170°C
	30/26	0.1 mm	120°C
		0.4 mm	130°C
	70/71	0.1 mm	140°C
		0.4 mm	170°C

1. Ambient temperature plus the temperature rise caused by current in the conductors and components.
2. FR-4 laminates should not be combined in one board with GPY prepregs.
3. When GPY laminates are combined in one board with FR-4 prepregs, the temperature **shall** be that specified for the FR-4 materials.
4. A multilayer board **shall** be limited to a maximum operating temperature for the total dielectric thickness shown above.
5. For reinforcement/resin combinations not specifically shown above, the maximum operating temperature should be close to that of a listed combination with the same resin.
6. Other materials in the same classification with higher operating temperatures may be available.
7. Dielectric thickness smaller than those shown above may be temperature rated by UL for certain laminate manufacturers.

4.3.2 Dielectric Thickness/Spacing The minimum dielectric thickness/spacing **shall** be specified on the master drawing. If the minimum dielectric spacing and the number of reinforcing layers are not specified, the minimum dielectric spacing is 0.09 mm and the number of reinforcing layers may be selected by the supplier.

Note: Minimum dielectric spacing may be specified to be 0.03 mm; however, low-profile copper foils should be used and the voltages employed should be taken into consideration so as not to cause breakdown between layers. See IPC-2221 for more information on electrical conductor spacing.

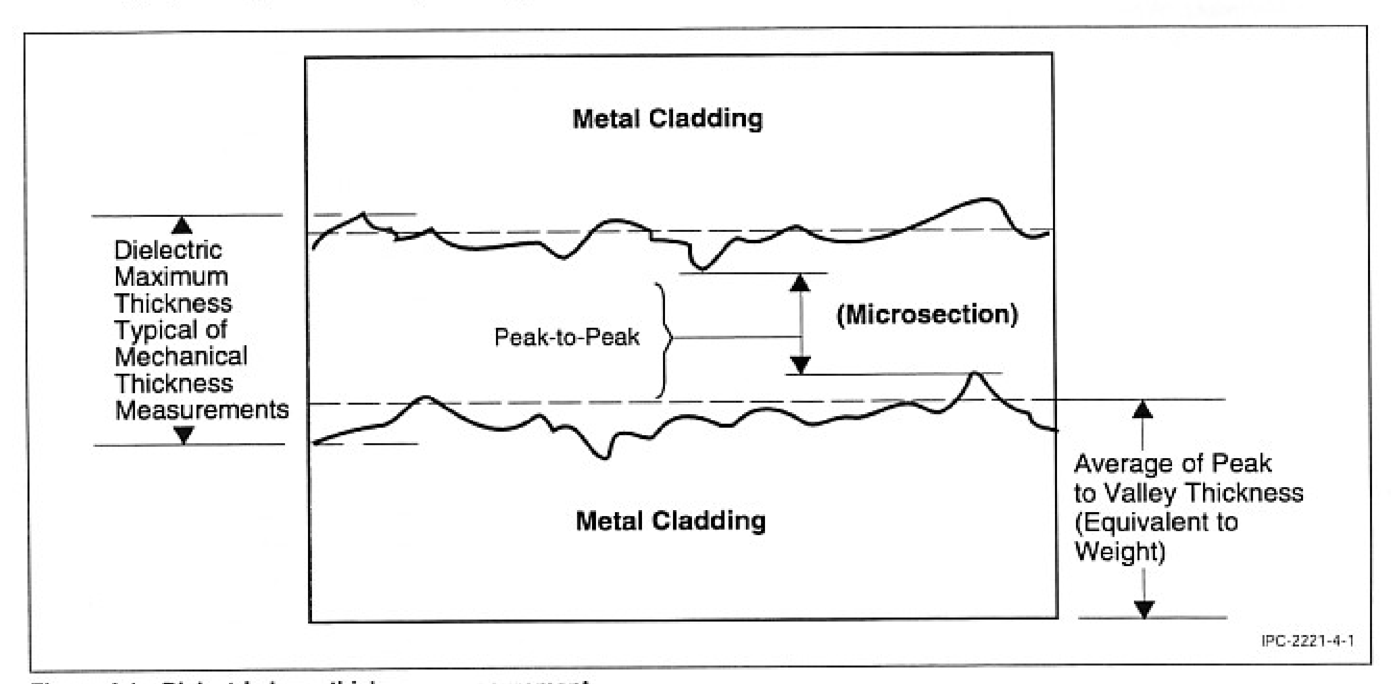


Figure 4-1 Dielectric layer thickness measurement

4.3.3 Laminate Properties

4.3.3.1 Thickness Tolerance When specifying overall multilayer board thickness, and individual dielectric thickness between layers, it is important to recognize the effects of accumulated tolerances of individual dielectrics on the overall thickness of completed boards.

4.3.3.2 Resin Content Laminates are a composite of resin and glass cloth or other reinforcement. As laminate thicknesses increase, heavier glass cloths or reinforcements are used, and the percentages of resin (resin content) decrease. Laminates with higher resin content tend to have higher coefficients of thermal expansion and lower dimensional stability. However, if resin contents are too low, weave exposure and measling may result. The glass to resin ratio of a laminate also has a direct effect on dielectric constant.

4.3.4 Prepreg

4.3.4.1 Epoxy Low T_g epoxies are the most common resin materials which are combined with glass cloth or other reinforcement and semi-cured to produce prepregs. Epoxy prepregs will produce acceptable product for Class 1, 2, and 3 multilayer printed boards. Because of different chemical processing requirements for various resin systems, it is preferred to use like resins for laminates and prepregs when constructing a multilayer printed board.

4.3.4.2 High-Temperature Prepregs High temperature prepregs include those made from resins such as Epoxy, Cyanate ester, Triazine blends and polyimide. High temperature prepregs may be used for specialized Class 2 multilayer printed board applications, but are more commonly used for Class 3 multilayer printed boards.

4.3.4.3 Glass Style A variety of glass cloth styles are available for prepregs (see IPC-EG-140). The glass cloth selection is dependent upon dielectric thickness and tolerance required, circuit filling needs, and electrical requirements of the dielectric.

4.3.4.4 Electrical Requirements For multilayer printed boards which have controlled impedance requirements, the dielectric constant of the laminated prepreg must be controlled. Because dielectric constant is a function of the resin/glass or other reinforcement ratio, prepreg styles should be chosen so that after lamination, the proper retained resin content is achieved in order to arrive at the specified dielectric constant.

4.3.5 Single-Clad Laminates Laminates with foil on one side may be used as an outer layer or internal layer of a multilayer printed board, as appropriate.

4.3.6 Double-Clad Laminates Laminates with foil on both sides may be used to provide either internal or external conductive layers. Double-clad laminate is specified in both industry and IPC specifications by the dielectric separation between the conductive layers as shown in Figure 4-1. Tables 4-2 through 4-6 provide information on the properties of finished bare laminates for different prepreg constructions. To establish final laminate thickness with copper, add 35 μm for each oz. of copper on the laminate.

4.3.7 Laminate Material Laminate materials **shall** be specified on the drawing. See Figure 4-2 for a mapping of the recommended material selection process.

Materials are generally purchased to meet the requirements of IPC-4101. A typical material code designation of a specific material would be L21 1500 C1/C1 A1A. When the finished product requires Underwriters Labs (UL) approval, material **shall** be ordered to meet UL specifications.

4.3.7.1 Typical Material Designation The first three characters of the code designate the type of material. "L" indicates laminate material. "P" indicates prepreg material.

- "L21"— Woven "E" glass fabric impregnated with flame resistant, epoxy resin of a type that is a majority of difunctional resin. Small amounts of multifunctional resin or novalacs are sometimes added to enhance the physical properties. This is the standard NEMA FR-4 grade manufactured by most laminators since the 1950s. The glass transition temperature (T_g) is normally from 110 to 150°C but not specified.
- "L25"— Woven "E" glass fabric impregnated with flame resistant epoxy resin which is commonly a polyfunctional type resin. This resin may be modified with other epoxies to increase the high temperature physical properties. This material may be used where repeated soldering operations to replace components are anticipated. The T_g is specified to be from 150 to 200°C.
- "L26"— This grade is similar to L25 except that the epoxy resin is modified with non-epoxy resins such as cyanate esters and/or bismaleimides. The uses are similar but where higher temperatures may be anticipated. The T_g is specified to be from 170 to 220°C.
- "L40"— Woven "E" glass fabric impregnated with polyimide resin. Introduced in the 1960s for high temperature operating environments such as missile engine controls, the resin has been supplied primarily from one European source. The natural color is opaque brown. The T_g is normally from 200 to 250° but not specified.
- "L42"— This grade is similar to L40 except that the polyimide resin may be modified with nonpolyimide resins. The primary purpose of the modifications are to improve the producibility of the printed board. The applications are similar. The T_g is specified as from 200 to 250°C.

Slash Sheet 1

FR-4 Copper Clad Laminate Construction Selection Guide

REG#	THICKNESS	CONSTRUCTION	% RC	DK	DK. TOL.	DS	Z CTE	THICK TOL	CHEM	MEASLE	AVAIL	COST	FLAT	SMOOTH	DRILL
FR-4 00	0.05mm	106	70	4.08	○	○	○	○	●	●	○	○		+	+
FR-4 01	0.07mm	1080	60	4.25	◓	●	○	○	●	●	○	●		+	+
FR-4 02	0.08mm	2x106	64	4.15	●	○	○	●	◓	●	○	○		+	+
FR-4 03	0.11mm	2x106	72	4.10	◓	●	○	◓	○	●	○	○		+	+
FR-4 04	0.11mm	2113	57	4.30	●	○	◓	●	○	○	○	●			+
FR-4 05	0.14mm	106/2113	56	4.30	●	●	◓	●	◓	○	●	○	−	+	+
FR-4 06	0.13mm	2x1080	59	4.25	○	⊗	◒	○	●	◓	◓	◓		+	+
FR-4 07	0.13mm	2116	53	4.40	○	○	●	○	○	○	○	●			
FR-4 08	0.16mm	106/2116	51	4.45	●	●	○	●	○	○	●	○	−		
FR-4 09	0.16mm	1080/2113	54	4.40	●	○	○	●	○	○	◓	○	−		−
FR-4 10	0.18mm	2x2113	50	4.50	○	○	○	◓	◓	○	○	○			+
FR-4 11	0.18mm	7628	40	4.75	○	○	●	◓	⊗	⊗	○	●		−	−
FR-4 12	0.21mm	2113/2116	50	4.50	●	●	○	●	○	○	●	○	−		
FR-4 13	0.21mm	2x2116	47	4.55	●	○	○	●	○	○	●	○	+		
FR-4 14	0.25mm	2x2116	52	4.4	◓	○	◓	◓	◓	◓	●	◓	+		−
FR-4 15	0.26mm	7628/1080	47	4.6	●	●	●	●	◒	○	◓	●	−	−	
FR-4 16	0.26mm	2x1080/2116	55	4.6	●	○	○	●	●	●	○	◒	+	+	
FR-4 17	0.31mm	2x1080/7628	47	4.6	●	●	◓	●	●	●	●	○	+	+	−
FR-4 18	0.32mm	7628/2116	47	4.6	●	◓	●	●	○	○	○	●	−	−	−
FR-4 19	0.37mm	2x7628	41	4.7	●	●	●	●	◒	○	●	●	+	−	−
FR-4 20	0.37mm	2x2113/7628	46	4.8	◓	●	◓	●	◓	●	◒	○	+		−
FR-4 21	0.43mm	2x2116/7628	48	4.5	●	●	◓	●	◓	◓	○	○			−
FR-4 22	0.43mm	2x7628/1080	43	4.7	●	●	●	●	◒	○	●	●	+	−	−
FR-4 23	0.48mm	2x7628/2116	43	4.8	●	●	◓	●	○	○	●	◓	+	−	−
FR-4 24	0.51mm	2x1080/2x7628	46	4.5	●	●	◓	●	●	●	○	○	+	+	−
FR-4 25	0.53mm	3x7628	40	4.7	●	●	●	●	○	◒	●	●	+	−	−
FR-4 26	0.64mm	2x2116/2x7628	47	4.6	●	●	◓	●	◓	●	●	●	+		−
FR-4 27	0.61mm	3x7628/1080	42	4.7	●	●	●	●	◒	○	●	●	−	−	−
FR-4 28	0.74mm	4x7628	41	4.7	●	●	●	●	○	○	●	●	+	−	−
FR-4 29	0.74mm	2x2113/3x7628	44	4.7	●	●	○	●	◓	◓	○	○	+		−
FR-4 30	0.75mm	4x7628/1080	42	4.7	●	●	◓	●	○	◒	◒	◓	+	−	−
FR-4 31	1.52mm	8x7628	42	4.7	●	●	◓	◓	○	○	●	◓	+	−	−

● Better + ◓ ○ Blank ◒ ⊗ Worse

IPC-Slash Sheet #1

Table 4-2 FR-4 Copper Clad Laminate Construction Selection Guide

Slash Sheet 2

High T_G FR-4 Copper Clad Laminate Construction Selection

REG#	THICKNESS	CONSTRUCTION	% RC	DK	DK. TOL.	DS	Z CTE	THICK TOL	CHEM	MEASLE	AVAIL	COST	FLAT	SMOOTH	DRILL
HFR-4 00	0.05mm	106	70	4.08	○	○	○	○	●	●	○	○		+	+
HFR-4 01	0.07mm	1080	60	4.25	◓	●	○	○	●	●	○	●		+	+
HFR-4 02	0.08mm	2x106	64	4.15	●	○	○	●	◓	●	○	○		+	+
HFR-4 03	0.11mm	2x106	72	4.10	◓	●	○	◓	○	●	○	○		+	+
HFR-4 04	0.11mm	2113	57	4.30	●	○	◓	●	○	○	○	●			+
HFR-4 05	0.14mm	106/2113	56	4.30	●	●	◓	●	◓	○	●	○	–	+	+
HFR-4 06	0.13mm	2x1080	59	4.25	○	⊗	◒	○	●	◓	◓	◓		+	+
HFR-4 07	0.13mm	2116	53	4.40	○	○	●	○	○	○	○	●			
HFR-4 08	0.16mm	106/2116	51	4.45	●	●	○	●	○	○	●	○	–		
HFR-4 09	0.16mm	1080/2113	54	4.40	●	○	○	●	○	○	◓	○	–		–
HFR-4 10	0.18mm	2x2113	50	4.50	○	○	○	◓	◓	○	○	○			+
HFR-4 11	0.18mm	7628	40	4.75	○	○	●	◓	⊗	⊗	○	●		–	–
HFR-4 12	0.21mm	2113/2116	50	4.50	●	●	○	●	○	○	●	○	–		
HFR-4 13	0.21mm	2x2116	47	4.55	●	○	○	●	○	○	●	○	+		
HFR-4 14	0.25mm	2x2116	52	4.4	◓	○	◓	◓	◓	◓	●	◓	+		–
HFR-4 15	0.26mm	7628/1080	47	4.6	●	●	●	●	◒	○	◓	●	–	–	
HFR-4 16	0.26mm	2x1080/2116	55	4.6	●	○	○	●	●	●	○	◒	+	+	
HFR-4 17	0.31mm	2x1080/7628	47	4.6	●	●	◓	●	●	●	●	○	+	+	–
HFR-4 18	0.32mm	7628/2116	47	4.6	●	◓	●	●	○	○	○	●	–	–	–
HFR-4 19	0.37mm	2x7628	41	4.7	●	●	●	●	◒	○	●	●	+	–	–
HFR-4 20	0.37mm	2x2113/7628	46	4.8	◓	●	◓	●	◓	●	◒	○	+		–
HFR-4 21	0.43mm	2x2116/7628	48	4.5	●	●	◓	●	◓	◓	○	○			–
HFR-4 22	0.43mm	2x7628/1080	43	4.7	●	●	●	●	◒	○	●	●	+	–	–
HFR-4 23	0.48mm	2x7628/2116	43	4.8	●	●	◓	●	○	○	●	◓	+	–	–
HFR-4 24	0.51mm	2x1080/2x7628	46	4.5	●	●	◓	●	●	●	○	○	+	+	–
HFR-4 25	0.53mm	3x7628	40	4.7	●	●	●	●	○	◒	●	●	+	–	–
HFR-4 26	0.64mm	2x2116/2x7628	47	4.6	●	●	◓	●	◓	●	●	●	+		–
HFR-4 27	0.61mm	3x7628/1080	42	4.7	●	●	●	●	◒	○	●	●	–	–	–
HFR-4 28	0.74mm	4x7628	41	4.7	●	●	●	●	○	○	●	●	+	–	–
HFR-4 29	0.74mm	2x2113/3x7628	44	4.7	●	●	○	●	◓	◓	○	○	+		–
HFR-4 30	0.75mm	4x7628/1080	42	4.7	●	●	◓	●	○	◒	◒	◓	+	–	–
HFR-4 31	1.52mm	8x7628	42	4.7	●	●	◓	◓	○	○	●	◓	+	–	–

● Better + ◓ ○ Blank ◒ ⊗ Worse

IPC-Slash Sheet #2

Table 4-3 High T_G FR-4 Copper Clad Laminate Construction Selection Guide

Slash Sheet 3

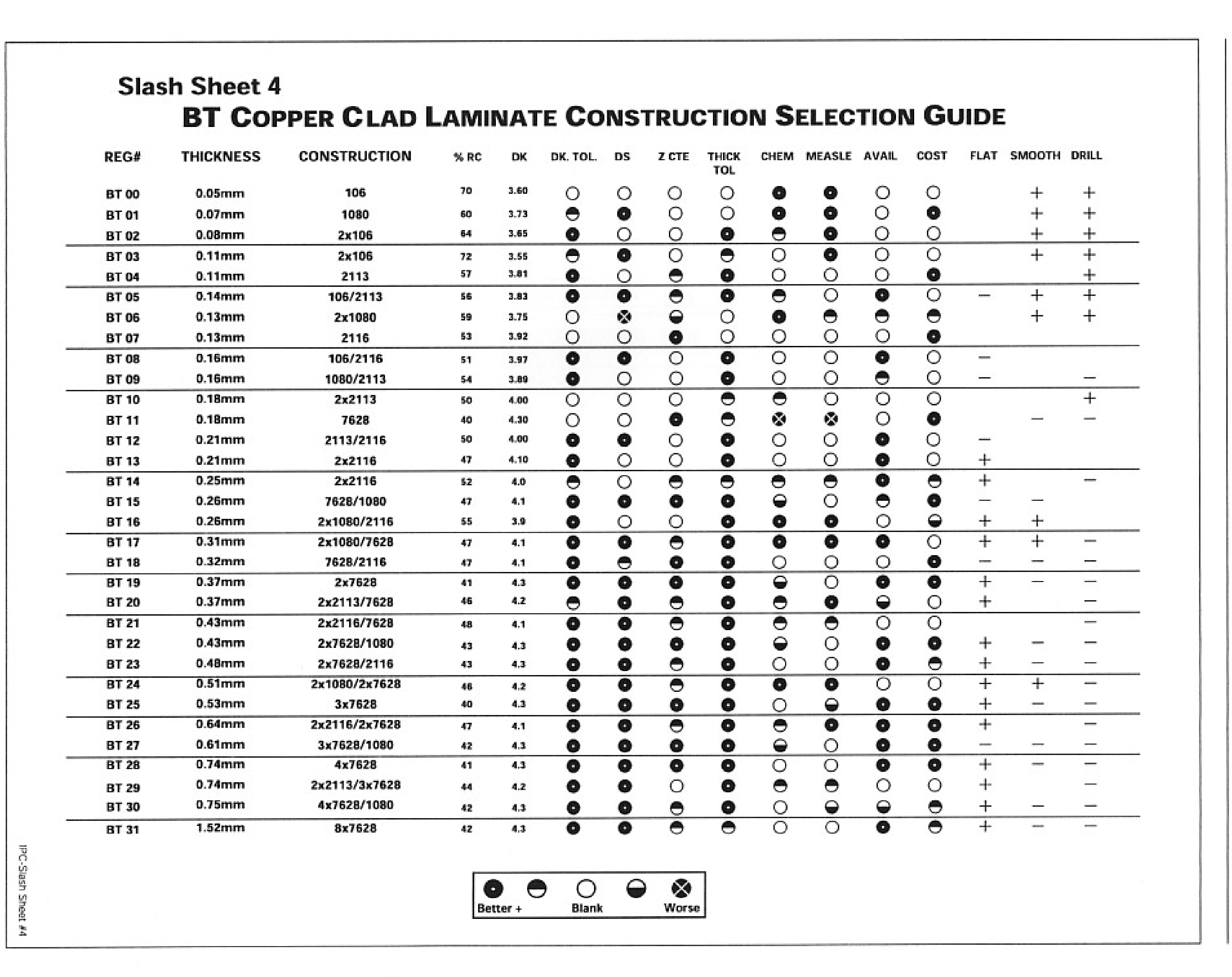
Cyanate Ester (170 to 230° Tg) Copper Clad Laminate Construction Selection Guide

REG#	THICKNESS	CONSTRUCTION	% RC	DK	DK. TOL.	DS	Z CTE	THICK TOL	CHEM	MEASLE	AVAIL	COST	FLAT	SMOOTH	DRILL
CE 00	0.05mm	106	70	3.10	○	○	○	○	●	●	○	○		+	+
CE 01	0.07mm	1080	62	3.37	◓	●	○	○	●	●	○	●		+	+
CE 02	0.08mm	2x106	60	3.44	●	○	○	●	◓	●	○	○		+	+
CE 03	0.11mm	2x106	68	3.28	◓	●	○	◓	○	●	○	○		+	+
CE 04	0.11mm	2113	54	3.70	●	○	◓	●	○	○	○	●			+
CE 05	0.14mm	106/2113	52	3.71	●	●	◓	●	◓	○	●	○	−	+	+
CE 06	0.13mm	2x1080	56	3.58	○	⊗	◒	○	●	●	◓	◓		+	+
CE 07	0.13mm	2116	52	3.71	○	○	●	○	○	○	○	●			
CE 08	0.16mm	106/2116	51	3.80	●	●	○	●	○	○	●	○	−		
CE 09	0.16mm	1080/2113	51	3.80	●	○	○	●	○	○	◓	○	−		−
CE 10	0.18mm	2x2113	50	3.78	○	○	○	◓	◓	○	○	○			+
CE 11	0.18mm	7628	40	4.08	○	○	●	◓	⊗	⊗	○	●		−	−
CE 12	0.21mm	2113/2116	48	3.85	●	●	○	●	○	○	●	○	−		
CE 13	0.21mm	2x2116	44	3.90	●	○	○	●	○	○	●	○	+		
CE 14	0.25mm	2x2116	52	3.71	◓	○	◓	◓	◓	◓	●	◓	+		−
CE 15	0.26mm	7628/1080	45	3.93	●	●	●	●	◒	○	◓	●	−	−	
CE 16	0.26mm	2x1080/2116	54	3.70	●	○	○	●	●	●	○	◒	+	+	
CE 17	0.31mm	2x1080/7628	45	3.93	●	●	◓	●	●	●	●	○	+	+	−
CE 18	0.32mm	7628/2116	44	3.90	●	◓	●	●	○	○	○	●	−	−	−
CE 19	0.37mm	2x7628	38	4.15	●	●	●	●	◒	○	●	●	+	−	−
CE 20	0.37mm	2x2113/7628	44	3.90	◓	●	◓	●	◓	●	◒	○	+		−
CE 21	0.43mm	2x2116/7628	44	3.90	●	●	◓	●	◓	◓	○	○			−
CE 22	0.43mm	2x7628/1080	39	4.11	●	●	●	●	◒	○	●	●	+	−	−
CE 23	0.48mm	2x7628/2116	39	4.11	●	●	◓	●	○	○	●	◓	+	−	−
CE 24	0.51mm	2x1080/2x7628	41	4.05	●	●	◓	●	●	●	○	○	+	+	−
CE 25	0.53mm	3x7628	38	4.15	●	●	●	●	○	◒	●	●	+	−	−
CE 26	0.64mm	2x2116/2x7628	44	3.90	●	●	◓	●	◓	◓	◓	○	+		−
CE 27	0.61mm	3x7628/1080	41	4.05	●	●	●	●	◒	○	●	●	−	−	−
CE 28	0.74mm	4x7628	38	4.15	●	●	●	●	○	○	●	●	+	−	−
CE 29	0.74mm	2x2113/3x7628	41	4.05	●	●	○	●	◓	◓	○	○	+		−
CE 30	0.75mm	4x7628/1080	38	4.15	●	●	◓	●	○	◒	◒	◓	+	−	−
CE 31	1.52mm	8x7628	40	4.08	●	●	◓	◓	○	○	●	◓	+	−	−

●	◓	○	◒	⊗
Better +		Blank		Worse

IPC-Slash Sheet #3

Table 4-4 Cyanate Ester (170 to 250° T_G) Copper Clad Laminate Construction Selection Guide



Slash Sheet 4

BT Copper Clad Laminate Construction Selection Guide

REG#	THICKNESS	CONSTRUCTION	% RC	DK	DK. TOL.	DS	Z CTE	THICK TOL	CHEM	MEASLE	AVAIL	COST	FLAT	SMOOTH	DRILL
BT 00	0.05mm	106	70	3.60	○	○	○	○	●	●	○	○		+	+
BT 01	0.07mm	1080	60	3.73	◓	●	○	○	●	●	○	●		+	+
BT 02	0.08mm	2x106	64	3.65	●	○	○	●	◓	●	○	○		+	+
BT 03	0.11mm	2x106	72	3.55	◓	●	○	◓	○	●	○	○		+	+
BT 04	0.11mm	2113	57	3.81	●	○	◓	●	○	○	○	●			+
BT 05	0.14mm	106/2113	56	3.83	●	●	◓	●	◓	○	●	○	−	+	+
BT 06	0.13mm	2x1080	59	3.75	○	⊗	◒	○	●	◓	◓	◓		+	+
BT 07	0.13mm	2116	53	3.92	○	○	●	○	○	○	○	●			
BT 08	0.16mm	106/2116	51	3.97	●	●	○	●	○	○	●	○	−		
BT 09	0.16mm	1080/2113	54	3.89	●	○	○	●	○	○	◓	○	−		−
BT 10	0.18mm	2x2113	50	4.00	○	○	○	◓	◓	○	○	○			+
BT 11	0.18mm	7628	40	4.30	○	○	●	◓	⊗	⊗	○	●		−	−
BT 12	0.21mm	2113/2116	50	4.00	●	●	○	●	○	○	●	○	−		
BT 13	0.21mm	2x2116	47	4.10	●	○	○	●	○	○	●	○	+		
BT 14	0.25mm	2x2116	52	4.0	◓	○	◓	◓	◓	◓	●	◓	+		−
BT 15	0.26mm	7628/1080	47	4.1	●	●	●	●	◒	○	◓	●	−	−	
BT 16	0.26mm	2x1080/2116	55	3.9	●	○	○	●	●	●	○	◒	+	+	
BT 17	0.31mm	2x1080/7628	47	4.1	●	●	◓	●	●	●	●	○	+	+	−
BT 18	0.32mm	7628/2116	47	4.1	●	◓	●	●	○	○	○	●	−	−	−
BT 19	0.37mm	2x7628	41	4.3	●	●	●	●	◒	○	●	●	+	−	−
BT 20	0.37mm	2x2113/7628	46	4.2	◓	●	◓	●	◓	●	◒	○	+		−
BT 21	0.43mm	2x2116/7628	48	4.1	●	●	◓	●	◓	◓	○	○			−
BT 22	0.43mm	2x7628/1080	43	4.3	●	●	●	●	◒	○	●	●	+	−	−
BT 23	0.48mm	2x7628/2116	43	4.3	●	●	◓	●	○	○	●	◓	+	−	−
BT 24	0.51mm	2x1080/2x7628	46	4.2	●	●	◓	●	●	●	○	○	+	+	−
BT 25	0.53mm	3x7628	40	4.3	●	●	●	●	○	◒	●	●	+	−	−
BT 26	0.64mm	2x2116/2x7628	47	4.1	●	●	◓	●	◓	●	●	●	+		−
BT 27	0.61mm	3x7628/1080	42	4.3	●	●	●	●	◒	○	●	●	−	−	−
BT 28	0.74mm	4x7628	41	4.3	●	●	●	●	○	○	●	●	+	−	−
BT 29	0.74mm	2x2113/3x7628	44	4.2	●	●	○	●	◓	◓	○	○	+		−
BT 30	0.75mm	4x7628/1080	42	4.3	●	●	◓	●	○	◒	◒	◓	+	−	−
BT 31	1.52mm	8x7628	42	4.3	●	●	◓	◓	○	○	●	◓	+	−	−

●	◓	○	◒	⊗
Better +		Blank		Worse

IPC-Slash Sheet #4

Table 4-5 BT Copper Clad Laminate Construction Selection Guide

Slash Sheet 5

Polyimide Copper Clad Laminate Construction Selection Guide

REG#	THICKNESS	CONSTRUCTION	% RC	DK	DK. TOL.	DS	Z CTE	THICK TOL	CHEM	MEASLE	AVAIL	COST	FLAT	SMOOTH	DRILL
POLY 00	0.05mm	106	72	4.00	○	○	○	○	●	●	○	◓		+	+
POLY 01	0.07mm	1080	63	4.02	◓	●	○	○	●	●	○	●		+	+
POLY 02	0.08mm	2x106	61	4.05	●	○	○	●	◓	●	○	○		+	+
POLY 03	0.11mm	2x106	59	3.89	◓	●	○	◓	○	●	○	○		+	+
POLY 04	0.11mm	2113	55	4.21	●	○	◓	●	○	○	○	●			+
POLY 05	0.14mm	106/2113	53	4.26	●	●	◓	●	◓	○	●	○	−	+	+
POLY 06	0.13mm	2x1080	57	4.15	○	⊗	◒	○	●	●	◓	◓		+	+
POLY 07	0.13mm	2116	54	4.23	○	○	●	○	○	○	○	●			
POLY 08	0.16mm	106/2116	52	4.28	●	●	○	●	○	○	●	○	−		
POLY 09	0.16mm	1080/2113	52	4.28	●	○	○	●	○	○	◓	○	−		−
POLY 10	0.18mm	2x2113	50	4.34	○	○	○	◓	◓	○	○	○			+
POLY 11	0.18mm	7628	39	4.66	○	○	●	◓	⊗	⊗	○	●		−	−
POLY 12	0.21mm	2113/2116	49	4.37	●	●	○	●	○	○	●	○	−		
POLY 13	0.21mm	2x2116	45	4.48	●	○	○	●	○	○	●	○	+		
POLY 14	0.25mm	2x2116	54	4.23	◓	○	◓	◓	◓	◓	●	◓	+		−
POLY 15	0.26mm	7628/1080	42	4.42	●	●	●	●	◒	○	◓	●	−	−	
POLY 16	0.26mm	2x1080/2116	56	4.18	●	○	○	●	●	●	○	◒	+	+	
POLY 17	0.31mm	2x1080/7628	47	4.42	●	●	◓	●	●	●	●	○	+	+	−
POLY 18	0.32mm	7628/2116	46	4.45	●	◓	●	●	○	○	○	●	−	−	−
POLY 19	0.37mm	2x7628	39	4.66	●	●	●	●	◒	○	●	●	+	−	−
POLY 20	0.37mm	2x2113/7628	45	4.48	◓	●	◓	●	◓	●	◒	○	+		−
POLY 21	0.43mm	2x2116/7628	45	4.18	●	●	◓	●	◓	◓	○	○			−
POLY 22	0.43mm	2x7628/1080	41	4.60	●	●	●	●	◒	○	●	●	+	−	−
POLY 23	0.48mm	2x7628/2116	40	4.63	●	●	◓	●	○	○	●	◓	+	−	−
POLY 24	0.51mm	2x1080/2x7628	42	4.57	●	●	◓	●	●	●	○	○	+	+	−
POLY 25	0.53mm	3x7628	39	4.66	●	●	●	●	○	◒	●	●	+	−	−
POLY 26	0.64mm	2x2116/2x7628	45	4.48	●	●	◒	●	◓	◓	◓	○	+		−
POLY 27	0.61mm	3x7628/1080	42	4.57	●	●	●	●	◒	○	●	●	−	−	−
POLY 28	0.74mm	4x7628	39	4.66	●	●	●	●	○	○	●	●	+	−	−
POLY 29	0.74mm	2x2113/3x7628	42	4.57	●	●	○	●	◓	◓	○	○	+		−
POLY 30	0.75mm	4x7628/1080	40	4.63	●	●	◓	●	○	◒	◒	◓	+	−	−
POLY 31	1.52mm	8x7628	41	4.60	●	●	◓	●	○	○	●	◓	+	−	−

● Better + ◓ ○ Blank ◒ ⊗ Worse

IPC-Slash Sheet #5

Table 4-6 Polyimide Copper Clad Laminate Construction Selection Guide

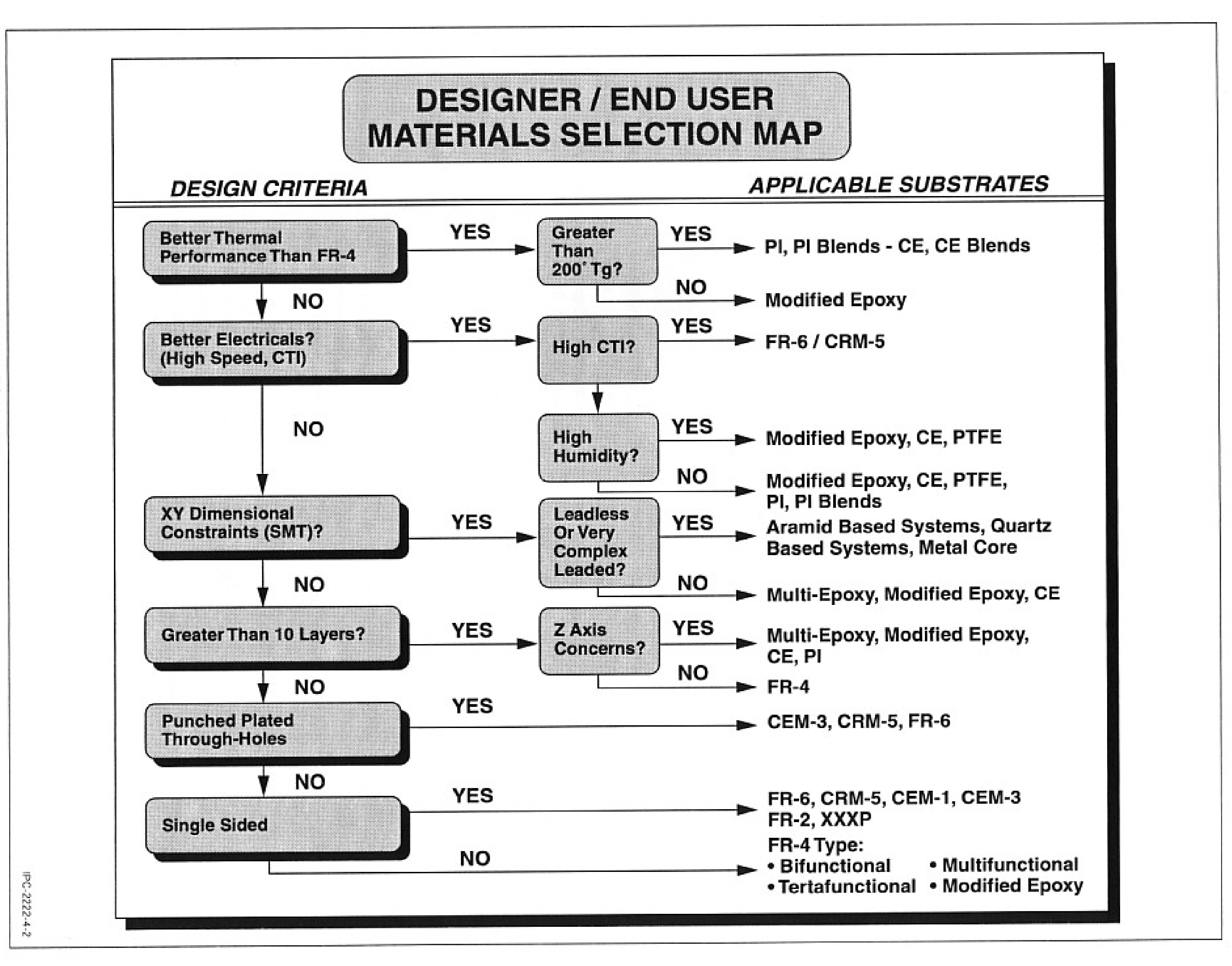


Figure 4-2 Designer / end user materials selection map

- "L41"— This grade is similar to L40 except that the resin chemistry may be altered. The applications are similar but higher operating temperature limits are possible. The T_g is specified as above 250°C.
- "L30"— Woven "E" glass fabric impregnated with BT resin. This resin is a mixture of bismaleimides and triazine. Its uses are similar to those for L25. The T_g is normally from 165 to 180°C. Hot strength retention should not be confused with flame resistance. Hot strength retention allows the board to operate at higher temperatures without losing electrical and mechanical properties. Flame resistance implies self-extinguishing properties.

4.3.7.2 Dielectric Thickness The next four numbers of the code (1500) designates the nominal dielectric thickness.

The nominal base thickness is identified by four digits that indicate the thickness of the base material in thousandths of a millimeter (i.e., 1500 represents a nominal base thickness of 1.5 mm). When English units are specified, the four digits indicate thickness in ten-thousandths of an inch [i.e., 0590 = 0.059 in]. The overall nominal thickness does not include the metal cladding. The nominal thickness (i.e., 1500) stands alone with no tolerance. The drawing title block tolerance of three decimal places does not apply. Later in the code is a designation that applies a tolerance to the nominal thickness per industry standards. Whatever nominal thickness (typically 1.5 mm) is required, the corresponding number is inserted.

4.3.7.3 Copper Foil Designation The type and nominal weight of the copper foil cladding is identified by the next five characters of the code (i.e., C1/C1). The first and fourth characters of this designator will consist of the following letters to indicate the type of copper foil cladding.

A — Copper, rolled, wrought (IPC-MF-150, Class 5).

B — Copper, rolled (treated).

C — Copper, drum side out, electrodeposited (IPC-MF-150, Class 1).

D — Copper, drum side out, (double treated) electrodeposited.

G — Copper, high ductility electrodeposited (IPC-MF-150), Class 2).

H — Copper, high temperature elongation (IPC-MF-150, Class 3).

J — Copper, annealed electrodeposited (IPC-MF-150, Class 4).

K — Copper, light cold rolled-wrought (IPC-MF-150, Class 6).

L — Copper, annealed-wrought (IPC-MF-150, Class 7).

O — Unclad

M — Copper, as rolled-wrought-low temperature (IPC-MF-150, Class 8).

N — Nickel

U — Aluminum

V — Copper-Invar-Copper (IPC-CF-152)

Type C or H copper foil claddings are most often used.

The second and the fifth characters of this designator will indicate the nominal copper foil weight in ounces per square foot (oz/ft^2). The two indicators, which are separated by a slash (third character), will use the actual numbers for copper foil 1 oz/ft^2 or over, and the following letters for copper foil under 1 oz/ft^2.

E — 0.125 oz/ft^2

Q — 0.25 oz/ft^2

T — 0.375 oz/ft^2

H — 0.50 oz/ft^2

M — 0.75 oz/ft^2

O — Unclad

X — For any weight or thickness not expressed (e.g., 10 oz. copper foil) by a single digit designator. For example, "C1/C1" designates 1 oz/ft^2 copper, drum side out, on one side and 1 oz/ft^2 copper drum side out, on the other side. The slash should be considered to be the base laminate.

Base materials that are unclad on both sides would be designated 00/00.

This designation does not mean the total amount of copper that should be on the surface after processing (see IPC-2221).

Copper foils can be specified in foil weights from 0.125 to 7 oz/ft^2.

"CX/00" is the requirement for single-sided boards.

"CX/CX" is the requirement for double-sided boards.

4.3.7.4 Pit Designation The thirteenth character ("A") in the material specification code denotes the class of pits and dents allowed in the copper foil. Class of foil indentations is determined by the total amount and individual length of the pits and dents. A "pit" is a disruption or void in the surface of the copper, and must be within the limitations allowed by the applicable procurement document. A "dent" is a depression in the surface of the base laminate, and under pressure during lamination, the dent is transferred to the surface of the copper. There are five allowable designations: "A", "B", "C", "D", and "X" (see IPC-4101).

- "A" designation [29 points per 300 mm x 300 mm area] is adequate down to 0.25 mm conductors and spaces.
- "B" designation [5 points per 300 mm x 300 mm area] should be considered below 0.25 mm conductors and spaces.

- "C" designation [17 points per 300 mm x 300 mm area].
- "D" designation [0 points per 300 mm x 300 mm area].
- "X" designation" **shall** be as agreed between user and supplier (IPC-4101).

4.3.7.5 Thickness Class Tolerance Designation The designation "1" is a thickness class tolerance specification for the base laminate (see table below).

- If the board is going to mount on standoffs and/or the thickness is of no importance, Class 1 should be specified.
- If the board is required to plug into an edge connector, Class 2 or even Class 3 should be considered. The finished board thickness includes the nominal laminate thickness and the class tolerance, plus all the additional plating thicknesses added together.

4.3.7.6 Bow and Twist Designation The final character in the material specification code (for the laminate material only, not the final etched board) is a "bow and twist" specification per the following. These values apply only to sheet sizes as manufactured, and to cut pieces having either dimension no less than 460 mm.

- For nominal thicknesses not shown, the bow or twist for the next lower thickness shown applies.
- A board that has both dimensions less than 460 mm generally requires an "A" specification.
- A board that has both dimensions greater than 460 mm requires a "B" specification.
- Boards that have one dimension less than 460 mm and the other greater than 460 mm, and have gold fingers on the long dimension, require a "B" specification.
- Class "X" indicates no bow or twist requirement and may be used only for single-sided boards. But in the case of bow, the percentage is stated in terms of the lateral dimension (length or width); in the case of twist, the percentage is stated in terms of the dimension from one corner to the diagonally opposite corner.

If the specification does not call out any class tolerances, the loosest requirements are assumed.

4.4 Conductive Materials Conductive materials **shall** be in accordance with the generic standard IPC-2221.

4.5 Organic Protective Coatings Organic protective coatings **shall** be in accordance with the generic standard IPC-2221.

4.6 Markings and Legends Marking and legends **shall** be in accordance with the generic standard IPC-2221.

5.0 MECHANICAL/PHYSICAL PROPERTIES

5.1 Fabrication Requirements Fabrication requirements **shall** be in accordance with the generic standard IPC-2221 and as follows:

Typical maximum fabrication panel size limits for board processing equipment are summarized in Table 5-1.

5.2 Product/Board Configuration Product/board configuration **shall** be in accordance with the generic standard IPC-2221 and as follows:

5.2.1 Board Geometries The following are considerations to be taken into account during the design of a printed board.

5.2.1.1 Borders and Spacing Borders and margins are commonly employed by the printed board fabricator to provide room for tooling features and other process control features (see Figure 5-1).

The size of such borders is usually in the range of 10 to 40 mm. This is determined by also taking into account an optimum number of boards per panel, obtaining optimum plating across the panel (especially important for high-density/fine line boards), etc.

When the printed boards are made using a print-and-etch procedure the border size may depend on the type of printed board being made, i.e., double-sided printed boards tend to have smaller borders; multilayer printed boards tend to have wider borders. Also, the size of the borders need not be the same on all four sides of the panel.

The margins between boards on a panel also have to accommodate the panel/board shearing, blanking and routing operations. Thus, their size is commonly chosen to be either 4.8, 5.0 and 6.5 mm or the nearest dimension suitable to maintain the board features on the basic processing grid.

5.2.1.2 Dimensional Aspect Ratio Board length to width relationships should be kept as similar as possible. Long narrow boards or unusually-shaped boards lead to excessive bowing/twisting. Dimensional instability, and associated problems at all stages of fabrication, assembly, test and system fixturing become factors in determining final board size.

5.2.2 Support Adequate mechanical support should be provided typically for at least two opposite edges of a printed wiring assembly. The location and method of support **shall** be such as to minimize shock and/or vibration to a level that will protect against fracturing or loosening of conductor foil, or breaking of the components or component leads as a result of flexing the printed board assembly within the tolerance of the applicable specification.

5.3 Assembly Requirements Assembly requirements **shall** be in accordance with the generic standard IPC-2221 and as follows:

Table 5-1 Panel Size to Manufacturing Operation Relationships

Operation	Typical Maximum Panel Size
Drill	460 mm x 610 mm
Scrub, deburr, and most conveyorized finishing equipment	610 mm x open
Plating equipment	Custom sized, check with fabricator
Exposure equipment	610 mm x 610 mm
Routing equipment	460 mm x 610 mm
Screening equipment	510 mm x 760 mm
Bare board test	460 mm x 460mm
Laminating press size (based on 610 mm x 760 mm press with 50 mm open area on plated edges)	510 mm x 660 mm
Solder coating	460 mm x 610 mm

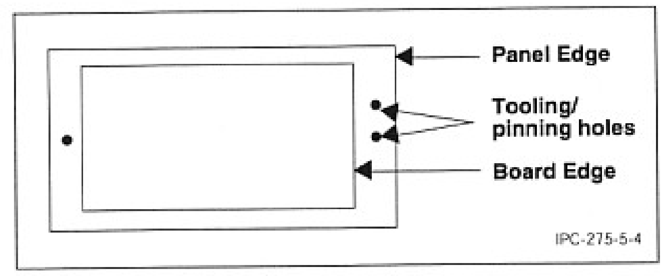


Figure 5-1 Panel borders

5.3.1 Assembly and Test Palletization of parts is a standard process in many instances for both test and assembly. This can be achieved using a number of different techniques. These include simple scoring, a combination of routing and scoring, and a combination of routing plus breakaway.

Scoring is the machining of a shallow, precise V-groove into the top and bottom surfaces of the laminate. It is generally accomplished using CNC equipment. As scoring allows the removal of rails and individual parts from a pallet, positional accuracy is critical. See Table 5-2 and Figure 5-2 for some standard scoring parameters.

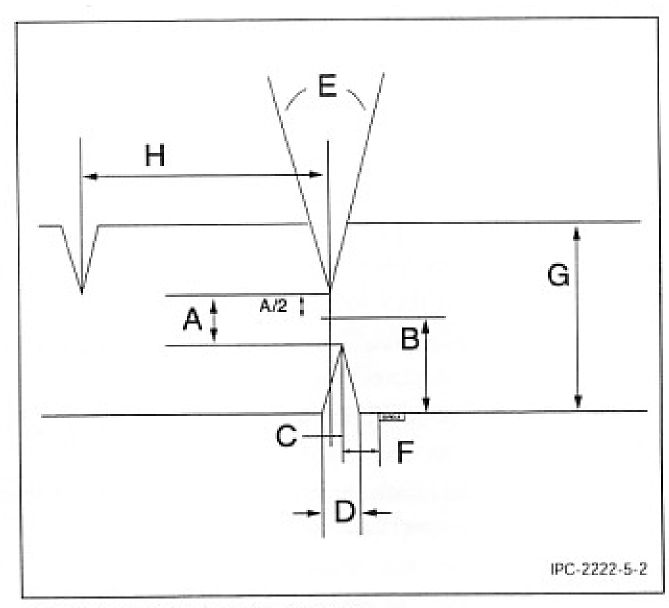


Figure 5-2 Scoring parameters

Table 5-2 Standard Scoring Parameters

Detail Letter	Title	Definition	Attainable Tolerances
A	Web	The material remaining between the two (2) 'V' scores on a plane perpendicular to the printed board surface	± 80 µm
B	Centrality	The distance the center of a web is offset from true center within the printed board.	± 80 µm
C	Blade offset	The distance the top and bottom scoring blades are offset from one another.	± 80 µm
D	Score width	The width of a score line at the surface of the printed board	± 80 µm
E	Cutter angle	The total angle of a scoring blade.	± 2°
F	Keep out area	The area, expressed from nominal score line placement, that no features should be placed within.	D/2 + All registration
G	Printed board thickness	Overall printed board thickness to be scored.	Per IPC standards
H	Trueness/ Position	The tolerance of two or more score lines on one side of the printed board. Measured from nominal, squareness and actual position.	± 80 µm cumulative

Routing is the process of profiling a pallet or printed board to the correct dimension using a cutting bit. This can be performed either by the use of a pin router and template, or a CNC routing machine.

Frequently a combination of routing and scoring is used, where both pallet and printed board are routed, leaving a small connection bridge. This bridge is then scored to facilitate removal following test and assembly.

The final method involves the use of routing and drilled breakaway tabs. Instead of scoring, a series of holes are drilled in the tab to facilitate removal. (See Figure 5-3.)

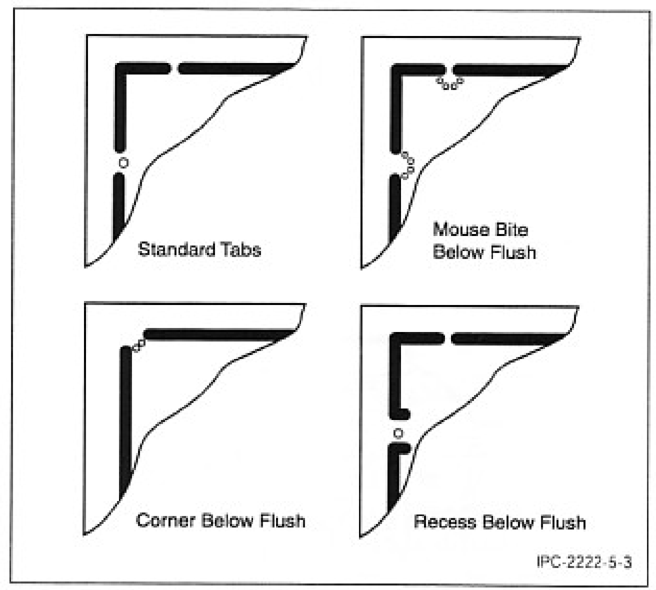


Figure 5-3 Breakaway tabs

5.4 Dimensioning Systems Dimensioning systems **shall** be in accordance with the generic standard IPC-2221 and as follows:

5.4.1 Grid Systems When manually designing printed boards, grid systems are used to locate components, plated-through holes, conductor patterns, and other features of the printed board and its assembly so they need not be individually dimensioned. When printed board features are required to be off a grid, they **shall** be individually dimensioned and toleranced on the master drawing.

Grid systems are always basic and have no tolerance, and therefore all features located on a grid **shall** be toleranced elsewhere on the master drawing. Grid systems **shall** be located with respect to a minimum of two printed board datums.

The grid increment **shall** be specified on the master drawing. The choice of grid increment is based on the component terminal location for through-hole components, and on the component center for surface mount components.

Typical grid increments are multiples of 0.13 mm for through-hole components, and 0.05 mm for surface mount components.

5.4.2 Profiles, Cutouts and Notches It is recommended that the number of cutouts and notches on the printed board be kept to a minimum in order to decrease the amount of time and effort necessary to fabricate them. This ultimately will help to minimize end-product printed board cost.

All such cutouts and notches must not interfere with the fabrication of other printed board/assembly features, such as the plating of edge-board contacts. The edges of internal cutouts **shall** be considered as external board edges and meet all requirements for hole and conductors to edge clearance (see 10.1.1). On multilayer printed boards the location of lands, clearances, ground planes and other conductive pattern features must also be considered.

For printed board shape routing, it is recommended that the cutouts and notches allow for a minimum of a 1.5 mm radius on internal corners (assuming that a 3 mm diameter router bit is used). Although a minimum internal radius of 0.75 mm can be obtained with a 1.5 mm diameter router bit, this requirement should be avoided as router efficiency and accuracy decreases substantially when smaller router bits are used. Recommended tolerances for the location and profile of cutouts and notches are shown in Table 5-3; however, the tolerances specified on the printed board drawing **shall** accommodate the dimensions and tolerances of the mating part.

Table 5-3 Tolerance of Profiles, Cutouts, Notches, and Keying Slots, as Machined, mm

Tolerances to be applied to profile of a surface:	Level A[1]	Level B	Level C
Profile feature	0.25	0.20	0.15
Location where greatest basic location dimension is less than 300.0	0.30	0.25	0.20
Location where greatest basic location dimension is greater than 300.0	0.35	0.30	0.25

[1]For definition of producibility level, see IPC-2221.

6.0 ELECTRICAL PROPERTIES

Electrical properties **shall** be in accordance with the generic standard IPC-2221.

7.0 THERMAL MANAGEMENT

Thermal management **shall** be in accordance with the generic standard IPC-2221.

8.0 COMPONENT AND ASSEMBLY ISSUES

Component assembly issues **shall** be in accordance with the generic standard IPC-2221 and as follows:

8.1 General Attachment Requirements In addition to the general attachment requirements outlined in the generic standard IPC-2221, the following **shall** apply:

8.1.1 Attachment of Wires/Leads to Terminals For cases in which more than one wire is attached to a terminal, the largest diameter wire should be mounted to the bottom-most post for ease of removal and repair. In general, no more than three attachments should be made to each section of a turret or bifurcated terminal. As an exception, bus bar terminals may hold more than three wires or leads per section when specifically designed to hold more.

8.1.2 Board Extractors Board extractors or handles are used to provide a convenient means of extracting the printed board from its mating connector. They are generally used where the amount of force makes it difficult to safely remove the board without damage to the electrical components or to the person removing the board.

Board extractors are commercially available and come in a variety of shapes and sizes.

Extractors are usually of the camming type and are mounted to the corners of the board. They provide a mechanical advantage for disengaging the connectors and a convenient place to grasp the board during removal.

Board extractors may be incorporated into the design of the board, or may require separate conditions in the printed board assembly. When board extractors are a part of the design, adequate reinforcement **shall** be used to properly allow the extracting action to remove the board from its connected assembly in the backplane (see Figure 8-1).

When board extractors are not a part of the printed board assembly, an extractor of the gripping variety may be used (see Figure 8-2). They grip the board in a particular area, which **shall** be kept free of components and circuitry. If a hook-type board extractor is used, where a hook passes through holes in the printed board, and then pulls the board out, special grommets should be used to reinforce the hole structure to avoid board crazing or cracking.

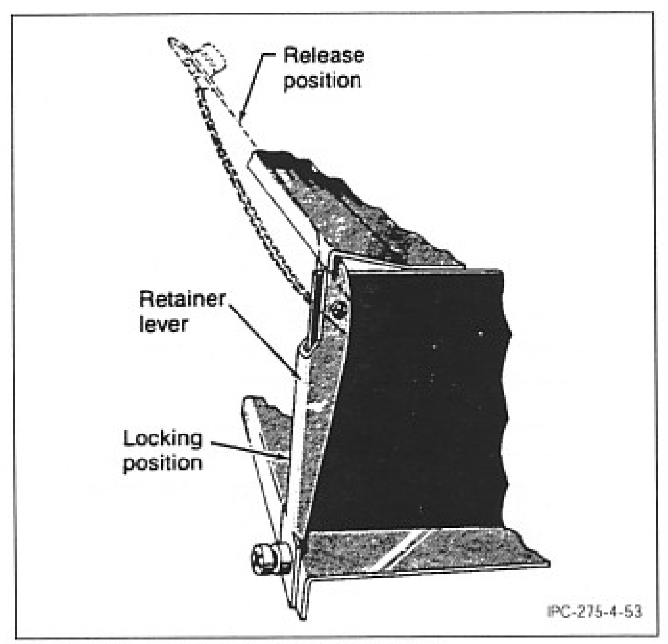


Figure 8-1 Permanent board extractor

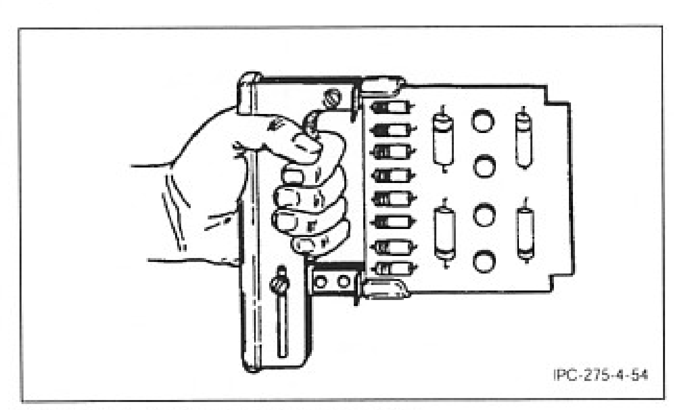


Figure 8-2 External board extractor

9.0 HOLE/INTERCONNECTIONS

9.1 General Requirements for Lands with Holes General requirements for lands with holes **shall** be in accordance with the generic standard IPC-2221 and as follows:

9.1.1 Land Requirements When eyelets or standoff terminals are used, the lands on external layers **shall** be so designed as to have a minimum a diameter of at least 0.5 mm greater than the maximum diameter of the projection of the eyelet of solder terminal flange.

9.1.2 Thermal Relief in Conductor Planes The relationship between the hole size, land and web area is critical. Typically, divide 60% of the minimum land area diameter by the number of webs desired to obtain the width of each web in accordance with the following example:

A. Land Size Calculation

Maximum hole size = 1.0 mm

Annular ring = 2 x 0.05 mm
= 0.10 mm

Fabrication allowance = 0.25 mm

Minimum land size = 1.0 mm + 0.10 mm + 0.25 mm
= 1.35 mm diameter

B. Thermal Relief Calculation

Total thermal width = 60% of land size
= 0.6 x 1.35 mm
= 0.80 mm

C. Original Web Size Calculation

2-web width = 1/2 of total thermal width
= 0.50 x 0.80 mm
= 0.40 mm

3-web width = 1/3 of total thermal width
= 0.33 x 0.80 mm
= 0.27 mm

4-web width = 1/4 of total thermal width
= 0.25 x 0.80 mm
= 0.20 mm

If the actual land diameter chosen is greater than the minimum value calculated, the percentage difference between the land diameters must be subtracted from the total web width calculation, i.e.:

Minimum land diameter = 1.35 mm

Actual land diameter = 1.70 mm

Percent difference = (1.70 - 1.35 mm)/1.35 mm
= 25%

New total web width = total web width
percent difference
= 0.80 mm - 25% (0.80 mm)
= 0.60 mm

D. Adjusted Web Size Calculation

2-web width = 1/2 of new total web width
= 0.50 x 0.60 mm
= 0.30 mm

3-web width = 1/3 of new total web width
= 0.33 x 0.60 mm
= 0.20 mm

4-web width = 1/4 of new total web width
= 0.25 x 0.60 mm
= 0.15 mm

Total cumulative copper web for all layers in any plated-through hole should not exceed 4.0 mm for 1 oz copper or 2.0 mm for 2 oz copper.

The total of the thermal relief cross-sectional area divided by the number of planes connected to the plated-through hole **shall** not violate current carrying capacity requirements for a given hole.

If the individual web width violates the intended minimum conductor width it **shall** be specified on the master drawing.

9.1.3 Clearance Area in Planes Clearance area in planes **shall** be provided in accordance with Figure 9-1.

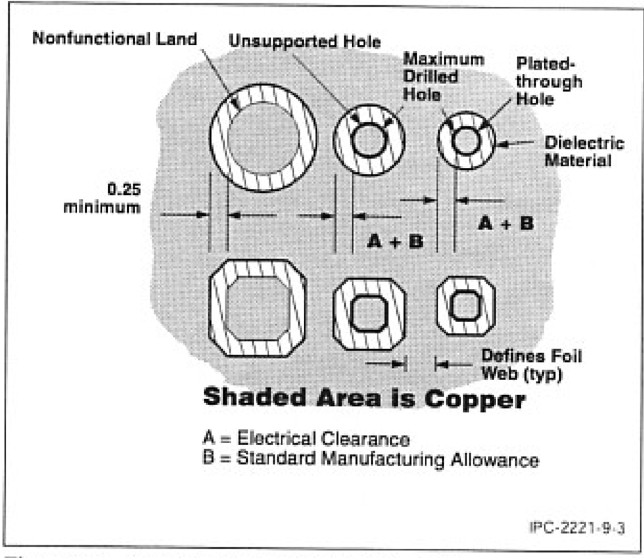


Figure 9-1 Clearance area in planes, mm

9.1.3.1 Small Pitch Clearance Area in Planes A special precaution must be observed when routing high speed circuits and/or very small pitch devices. When routing small pitch devices and/or vias routed on small grids, designers must remain aware of factors relating to power/ground plane clearance areas. When the pitch is very small, designers must remain cognizant of the narrow foil web between clearance openings (see Figure 9-2). As the clearance area (diameter) is made larger, the foil web between clearance areas becomes smaller. Designs having very small foil webs are less desirable because of their reduced current carrying capability, potential for increased voltage drop, higher EMI emissions, and reduced thermal dissipating characteristics. It is highly desirable for heat generating devices to be "heatsinked" down through via holes and dissipated across the surface of inner planes. For these reasons the foil web should be as large as possible and overlapping clearance areas in planes should be avoided. Use the following formula when dealing with small pitch devices and/or via holes, when routing on very small grids.

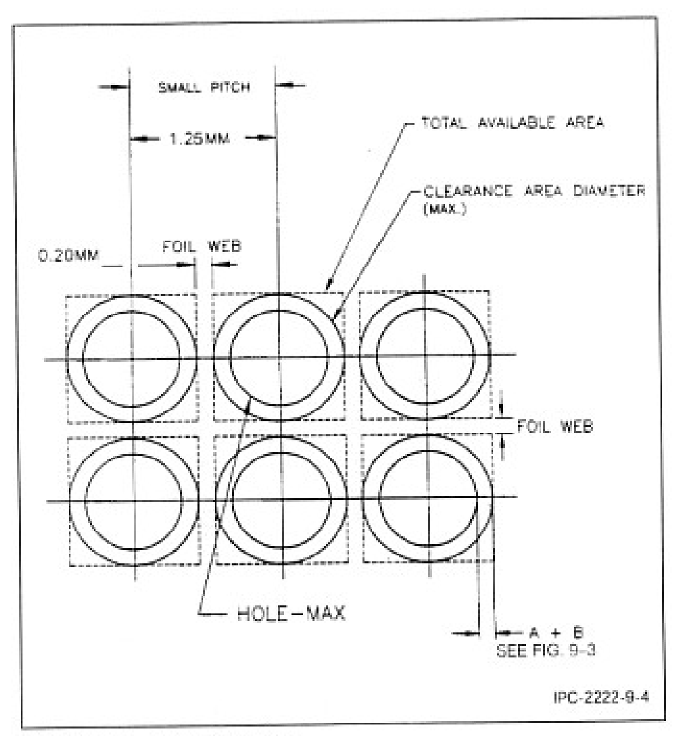


Figure 9-2 Foil web size

Typical example:
Determine the desired width of foil web.
Ex: 0.20 mm Web

Total available area = pitch - foil web
= 1.25 - 0.20 mm
= 1.05 mm

B = 0.25 mm min. fabrication allowance per Table 9-1 of IPC-2221.
Clearance area (diameter) = hole diameter (max.) + 2B
= 0.35 mm + 2 (0.25 mm)
= 0.35 mm + 0.50 mm
= 0.85 mm

Note: Maximum clearance area (diameter) in the plane is calculated by using the formula above and must not exceed the total available area.

9.1.4 Nonfunctional Lands Nonfunctional lands should be included on internal layers for all plated-through holes. Nonfunctional lands need not be used where electrical clearance requirements do not permit, such as ground planes, voltage planes and thermal planes. For high layer count boards, greater than 10 layers, it is recommended to remove some of the nonfunctional lands in the vertical stack. Plated-through holes passing through internal conductive planes (ground, voltage, etc.) and thermal planes **shall** meet the same minimum spacing requirements as conductors on internal layers, and should meet the minimum spacing requirements of Figure 9-2.

9.1.5 Conductive Pattern Feature Location Tolerance The presentation in Table 9-1 is for the tolerance to be applied to the nominal dimension chosen for the location of the lands connector contacts and conductors in relation to the datums. This tolerance includes tolerances for master pattern accuracy, material movement, layer registration and fixturing.

Table 9-1 Feature Location Tolerances (Lands, Conductor Pattern, etc.) (Diameter True Position)

Greatest Board/ X, Y Dimension	Level A	Level B	Level C
Up to 300 mm	0.30 mm	0.20 mm	0.10 mm
Up to 450 mm	0.35 mm	0.25 mm	0.15 mm
Up to 600 mm	0.40 mm	0.30 mm	0.20 mm

Note: Conductor pattern registration may be expressed in terms of minimum annular ring violation, which establishes manufacturing registration allowances.

9.2 Holes Holes **shall** be in accordance with the generic standard IPC-2221 and as follows:

9.2.1 Unsupported Holes

9.2.1.1 Diameter of Unsupported Holes When using the basic dimensioning system, holes shall be expressed in terms of maximum material (MMC) and least material condiction (LMC) limits. The diameter of an unsupported component hole **shall** be such that the MMC of the lead subtracted from the MMC of the hole provides a clearance between a minimum of 0.15 mm and a maximum of 0.5 mm. The number of different hole sizes **shall** be kept to a minimum. When flat ribbon leads are mounted through unsupported holes, the difference between the nominal diagonal of the lead and the inside diameter of the unsupported hole **shall** not exceed 0.5 mm and **shall** be not less than 0.15 mm.

9.2.1.2 Unsupported Hole Tolerance When using the basic dimensioning system, holes **shall** be expressed in terms of maximum material condition (MMC) and least material condition (LMC) limits. The bilateral tolerances shown in Table 9-2 are used to determine the MMC-LMC limit for the appropriate hole diameter; thus, a hole 1.0 ± 0.05 mm would be expressed as 0.95-1.05 mm.

Table 9-2 Minimum Unsupported Holes Tolerance Range (Difference between high and low hole size limits)

Hole Diameter	Level A	Level B	Level C
0.1 - 0.8 mm	0.15 mm	0.10 mm	0.05 mm
0.81 - 1.6 mm	0.20 mm	0.15 mm	0.10 mm
1.61 - 5.0 mm	0.30 mm	0.20 mm	0.15 mm

9.2.1.3 Eyelet Hole Diameter When eyelets are used, the diameter of holes in which eyelets are inserted **shall** not exceed the outside diameter of the barrel of the eyelet by

more than 0.15 mm. The relationships between maximum and minimum barrel diameters and wire diameters **shall** be as shown in Table 9-3.

9.2.2 Plated-Through Holes The maximum and minimum plated-through hole diameters used to attach component leads or pins to the printed board **shall** be evaluated in accordance with Table 9-3. Both minimum and maximum leads **shall** be taken into consideration in evaluating the finished plated-through hole requirements. If the lead is a ribbon lead, the minimum and maximum diagonal of a flat ribbon lead **shall** be considered. Table 9-3 shows the limits of the plated-through hole.

These limits **shall** be optimized so that manufacturability is enhanced to provide the most liberal tolerances allowable (see Figure 9-3).

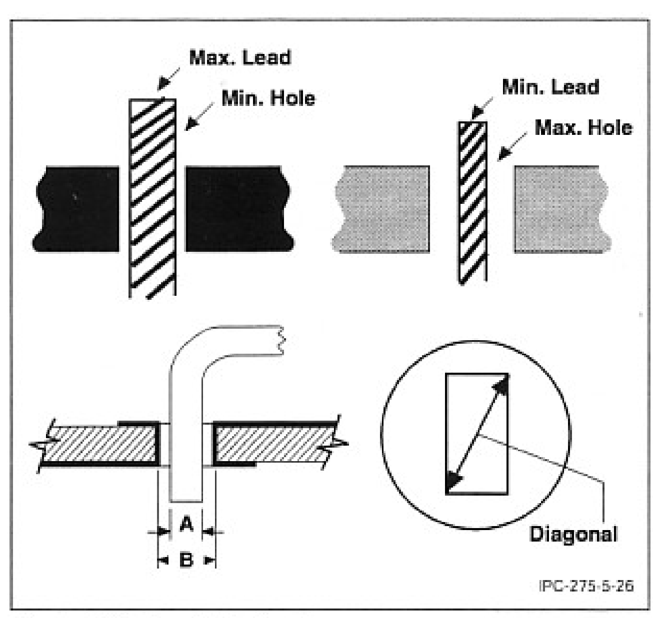


Figure 9-3 Lead-to-hole clearance

Unless otherwise specified, the hole size **shall** be the finished plated size after solder coating or final plating and fusing, if required. The hole size **shall** be specified on the master drawing. Plated-through holes used for functional interfacial connections **shall** not be used for the mounting of devices which put the plated-through hole in compression. Plated-through holes used for functional interfacial connections **shall** not be used for the mounting of eyelets, solder terminals, or rivets. Plated-through holes **shall** be used for all interfacial connections on multilayer boards Type 3 through Type 6 (inclusive). Platings and coatings **shall** be in accordance with IPC-2221.

9.2.2.1 Aspect Ratio The aspect ratio of plated-through holes plays an important part in the ability of the manufacturer to provide sufficient plating within the plated-through hole, as well as in the reliability of the PTH/PTV structure (see IPC-D-279). Table 9-4 provides information on the producibility of aspect ratios for various levels of complexity.

9.2.2.2 Plated-Through Hole Tolerances When using the basic dimensioning system, plated-through holes used to attach component leads or pins to the printed board should be expressed in terms of MMC and LMC limits. The bilateral tolerances shown in Table 9-5 are used to determine the MMC-LMC limit for the appropriate finished hole diameter. Thus, a hole 1.0 ± 0.05 mm would be expressed as 0.95-1.05 mm. When hole size is less than one-fourth the basic board thickness, the tolerance **shall** be increased by 0.05 mm.

9.2.2.3 Minimum Hole Sizes for Plated-Through Hole Vias In order to meet the performance requirements of the various classes of equipments, the plated-through hole size to board thickness aspect ratio for plated-through hole vias should be in accordance with Table 9-4. Table 9-6 provides information on the minimum drilled hole to be used in conjunction with various board thicknesses. The table reflects the three classes of equipment assuming that each class requires a slightly more severe environment, thus having to meet more stringent thermal cycling conditions (see IPC-D-279).

If a particular class requires more stringent cycling than shown in the table, the user may invoke the requirements of a larger drilled hole size.

Solder may fill the through hole or via if processed with fused tin-lead plate or solder coating. Partial filling at assembly can create stress concentrations affecting reliability. For drilled hole diameters 0.35 mm or less and aspect ratios of 4:1 or larger, the fabricator should mask or plug by a suitable method the plated through vias to prevent entry of solder.

The drilled hole size used for through hole vias **shall** be represented on the master drawing as the maximum plated-through hole dimension that assumes that the hole contains a minimum plating thickness. No minimum plated-through hole dimension should be specified since the through-hole via contains no component lead or pin and could theoretically be plated shut. Thus, a master drawing call out of 0.0-0.2 mm diameter reflects drilling a 0.25 mm hole that can contain a minimum plating of 0.025 mm to a maximum plating of 0.125 mm copper per side.

In addition, Table 9-6 contains a letter code in each of the boxes. This letter code reflects the producibility level of complexity in each of the particular hole size to aspect ratios.

9.2.3 Etchback Etchback, when required, will reduce the annular ring support on internal layers of the board. Therefore, this should be taken into account when specifying plated-through hole land size. However, the maximum

Table 9-3 Plated-Through Hole Diameter to Lead Diameter Relationships

Lead Diameter	Level A	Level B	Level C
Maximum hole to minimum lead diameter	No greater than 0.7 mm over minimum lead diameter	No greater than 0.7 mm over minimum lead diameter	No greater than 0.6 mm over minimum lead diameter
Minimum hole to maximum lead diameter	No less than 0.25 mm over maximum lead diameter	No less than 0.20 mm over maximum diameter	No less than 0.15 mm over maximum lead diameter

Table 9-4 Plated-Through Hole Aspect Ratio

	Level A	Level B	Level C
Aspect Ratios	≤5:1	6:1 to 8:1	9:1 and up

etchback allowed on the master drawing **shall** not be greater than the minimum design annular ring.

9.3 Drill Size Recommendations for Printed Boards Drill sizes as related to maximum board thickness are shown in Table 9-7. These drills may be used to drill unsupported or plated-through holes in rigid printed boards.

Although the designer rarely specifies the drilled hole size, these dimensions may be used in determining minimum annular ring calculations, or minimum land sizes.

Also shown in Table 9-7 are the maximum board thicknesses to which the various drill sizes should be applied. (Note: These dimensions are maximums. Thinner boards may be used to accommodate all drills shown within a drill set category.) Designers are encouraged to limit the number of drill selections to approximately 10 drill sizes per each printed board. Where possible a lesser number should be used, recognizing that the manufacturer needs certain drill sizes for tooling hole and other configurations. It is recommended that the designer select no more than one drill size from any given row in order to optimize drill size operation. The information in Table 9-7 is a suggested guideline based on the availability of existing manufacturing drills.

Table 9-7 Drill Size Recommendations Related to Maximum Board Thickness

Drill size (mm)	Maximum Board Thickness (mm), as drilled
≤0.10	≤0.25
0.15-0.25	≤1.0
0.30-0.45	≤1.6
0.50-0.85	≤3.2
0.90-1.05	≤4.8
≤1.10	≤6.4

10.0 GENERAL CIRCUIT FEATURE REQUIREMENTS

10.1 Conductor Characteristics Conductor characteristics **shall** be in accordance with the generic standard IPC-2221 and as follows:

10.1.1 Edge Spacing Except for edge-board contacts, the minimum distance between conductive surfaces and the edge of the finished board, or a non-plated through hole, **shall** not be less than the minimum spacing specified in Table 6-1 of IPC-2221 plus 0.4 mm. Printed boards that slide into guides **shall** have a minimum external conductor to guide distance of 1.25 mm or minimum electrical clearance (see Table 6-1 of IPC-2221), whichever is greater. Special design applications in areas such as high voltage, surface mount, and radio frequency (RF) technology may require variances to these requirements. Ground and heat sink planes may extend to the edge when required by design.

Table 9-5 Minimum Plated-Through Hole Diameter Tolerance Range, mm (Difference between high and low hole size limits)

Hole Diameter	Level A	Level B	Level C
0.1 to 0.8	0.20	0.15	0.10
>0.8 to 1.6	0.30	0.20	0.10
>1.6 to 5.0	0.40	0.30	0.20

Table 9-6 Minimum Drilled Hole Size for Plated-Through Hole Vias

Board Thickness	Class 1	Class 2	Class 3
<1.0 mm	Level C 0.15 mm	Level C 0.2 mm	Level C 0.25 mm
1.0 mm to 1.6 mm	Level C 0.2 mm	Level C 0.25 mm	Level B 0.3 mm
1.6 mm to 2.0 mm	Level C 0.3 mm	Level B 0.4 mm	Level B 0.5 mm
>2.0 mm	Level B 0.4 mm	Level A 0.5 mm	Level A 0.6 mm

Note: If copper plating thickness in hole is greater than 0.03 mm, hole size can be reduced by one class.

10.1.2 Balanced Conductors Whenever possible, to reduce bow and twist and to increase dimensional stability, conductors should be balanced within an individual layer. Conductor routing density should be spread throughout the board wherever possible, to avoid the need for special etching or plating thieves.

Plating thieves are added metallic areas which are nonfunctional within the finished board profile but allow uniform plating density, giving uniform plating thickness over the board surface.

The multilayer printed board structure should also be as balanced as possible providing equal layers of signal conductors and planes to either side of the center of the multilayer construction.

10.1.3 Flush Conductors for Rotating or Sliding Contacts When flush circuits are required for application as mating contact surfaces, the degree of flushness **shall** meet the requirements of Table 10-1. Figure 10-1 (A and B) shows a typical flush circuit design.

Note: The level of flushness required is normally a function of factors related to the mating contact, such as contact size, shape, load, rotational force, surface friction, etc. It is recommended that the designer fully understand the dynamics of the mating contact system before establishing surface flushness requirements.

Table 10-1 Surface Flushness Requirements

Level A	Level B	Level C
As agreed between user and supplier	± 0.013 mm	± 0.005 mm

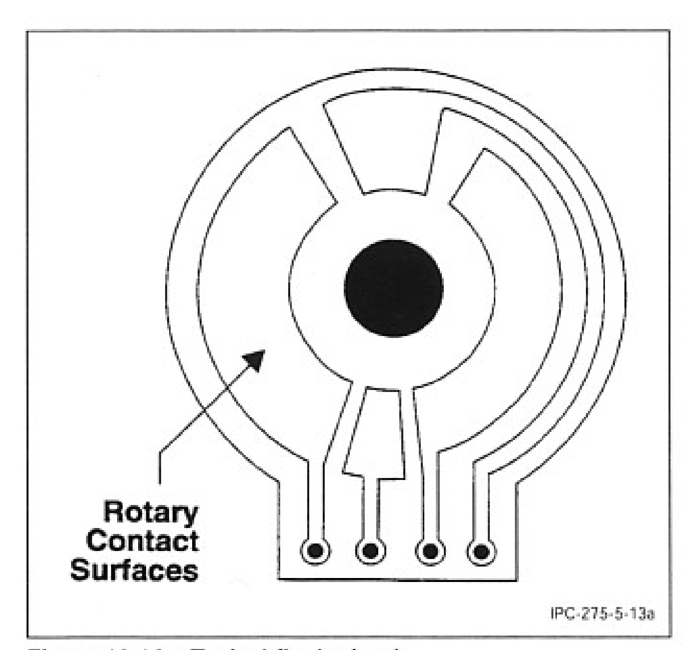


Figure 10-1A Typical flush circuit

10.1.4 Metallic Finishes for Flush Conductors Metallic finishes for flush conductor contacts should be gold over nickel, or other suitable corrosion resistant, low contact resistance finish.

10.2 Land Characteristics Land characteristics **shall** be in accordance with the generic standard IPC-2221 and as follows:

10.2.1 Lands for Interfacial Connection Vias Lands for interfacial connections **shall** meet the requirements of 10.1.

10.2.2 Offset Lands Lands, when used in conjunction with clinched leads, may be located adjacent to (not surrounding) the lead termination hole. The land **shall** be a sufficient distance from the hole to allow clipping of the part lead prior to unsoldering the part lead from the land.

10.2.3 Conductive Pattern Feature Location Tolerance The presentation in Table 9-1 is for the tolerance to be applied to the nominal dimension chosen for the location of the lands connector contacts and conductors in relation to the datums. This tolerance includes tolerances for master pattern accuracy, material movement, layer registration and fixturing. (See IPC-SM-782.)

10.2.4 Nonfunctional Lands See 9.1.4.

10.3 Large Conductive Areas Large conductive areas (Planes 3 mm wide or larger conductors) increase the likelihood for blistering, warping, or heat shielding during wave or reflow soldering.

Large areas that cover more than a 25.0 mm diameter may be broken up into a cross-hatched or similarly patterned area (see Figure 10-2). Modifications to large conductive areas shall not adversely impact the electrical characteristics or performance of the board. Large conductive areas should not be on the solder side of the board unless solder resist or similar is used.

External conductors that extend beyond a 25.0 mm diameter circle should contain etched areas that break upthe large conductive area, but retain the continuity and functionality of the conductor. If etched areas are not provided, other methods should be used to minimize blistering or bowing.

Large conductive areas should, if possible, be on the primary side of the board. If solder resist is employed over meltable metals, conductive areas wider than 1.3 mm **shall** not be employed under the solder resist coatings.

When a conductive area that extends beyond a 25.0 mm diameter is used on a internal layer, the conductive area

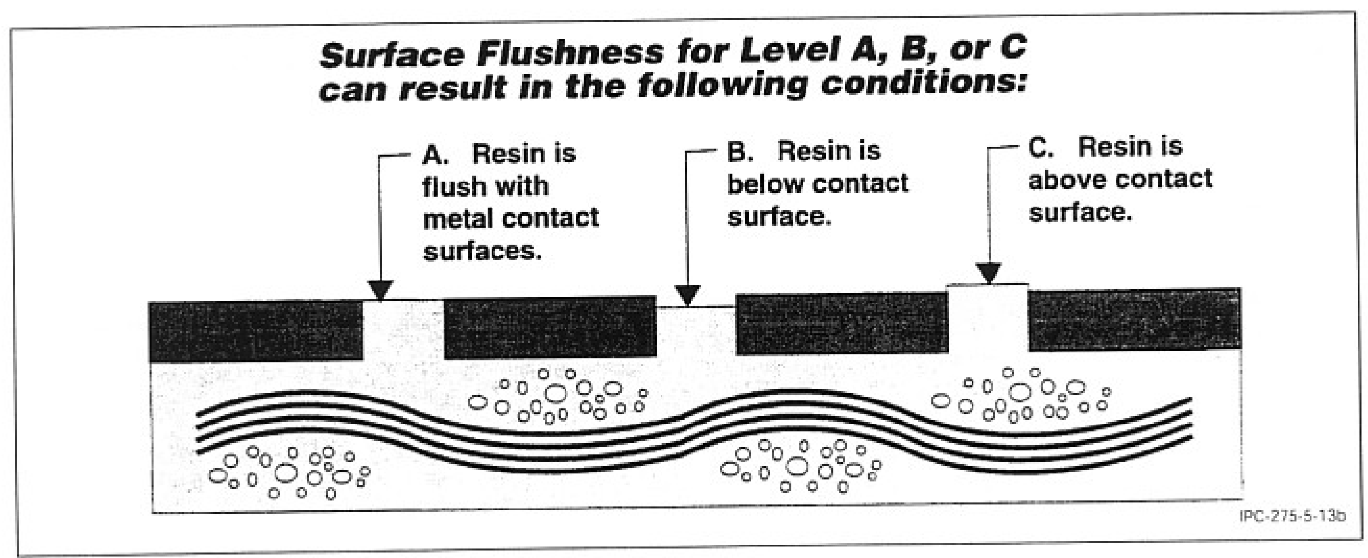


Figure 10-1B Surface flushness conditions

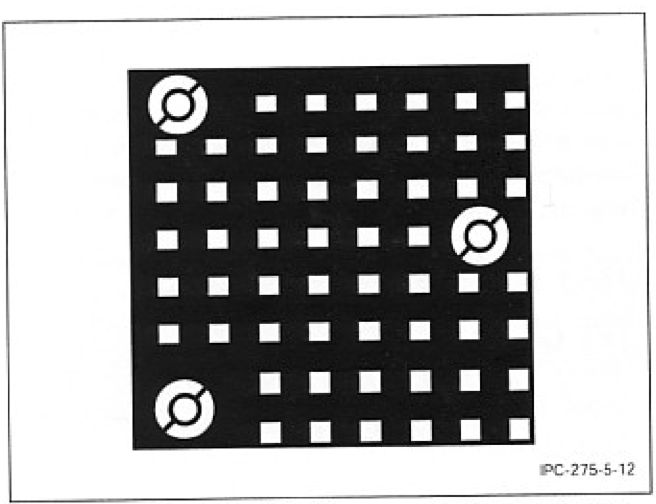


Figure 10-2 Cross-hatched large conductive layers with isothermal conductors

should be placed as near to the center or the board as possible, and should contain etched areas that will break up the large conductive area, but will retain the continuity and functionality of the conductor. When UL requirements are imposed, the area **shall** be within the limits approved by UL for the printed board manufacturer.

11.0 DOCUMENTATION

Documentation **shall** be in accordance with the generic standard IPC-2221 and as follows:

11.1 Filled Holes Via holes designated to be filled by the design requirements **shall** be identified on the drawing.

11.2 Nonfunctional Holes Electrically nonfunctional supported holes should be identified as such on the fabrication or assembly drawing and do not need to meet the electrical connection requirement.

12.0 QUALITY ASSURANCE

Quality Assurance **shall** be in accordance with the generic standard IPC-2221.

INDEX

The following is an index of key subjects related to specific paragraph numbers in this standard. The index is organized alphabetically.



ANSI/IPC-T-50 Terms and Definitions for Interconnecting and Packaging Electronic Circuits Definition Submission/Approval Sheet

The purpose of this form is to keep current with terms routinely used in the industry and their definitions. Individuals or companies are invited to comment. Please complete this form and return to:

IPC
2215 Sanders Road
Northbrook, IL 60062-6135
Fax: 847 509.9798

SUBMITTOR INFORMATION:

Name: ____________________

Company: ____________________

City: ____________________

State/Zip: ____________________

Telephone: ____________________

Date: ____________________

❑ This is a **NEW** term and definition being submitted.
❑ This is an **ADDITION** to an existing term and definition(s).
❑ This is a **CHANGE** to an existing definition.

Term	Definition

If space not adequate, use reverse side or attach additional sheet(s).

Artwork: ❑ Not Applicable ❑ Required ❑ To be supplied
❑ Included: Electronic File Name: ____________________

Document(s) to which this term applies: ____________________

Committees affected by this term: ____________________

Office Use

IPC Office

Date Received: ____________________
Comments Collated: ____________________
Returned for Action: ____________________
Revision Inclusion: ____________________

Committee 2-30

Date of Initial Review: ____________________
Comment Resolution: ____________________
Committee Action: ❑ Accepted ❑ Rejected
❑ Accept Modify

IEC Classification

Classification Code • Serial Number

Terms and Definition Committee Final Approval Authorization:

Committee 2-30 has approved the above term for release in the next revision.

Name: ____________________ Committee: IPC 2-30 Date: ____________________

Technical Questions

The IPC staff will research your technical question and attempt to find an appropriate specification interpretation or technical response. Please send your technical query to the technical department via:

tel 847/509-9700
fax 847/509-9798
www.ipc.org
e-mail: answers@ipc.org

IPC World Wide Web Page www.ipc.org

Our home page provides access to information about upcoming events, publications and videos, membership, and industry activities and services. Visit soon and often.

IPC Technical Forums

IPC technical forums are opportunities to network on the Internet. It's the best way to get the help you need today! Over 2,500 people are already taking advantage of the excellent peer networking available through e-mail forums provided by IPC. Members use them to get timely, relevant answers to their technical questions. Contact KeachSasamori@ipc.org for details. Here are a few of the forums offered.

TechNet@ipc.org

TechNet forum is for discussion of issues related to printed circuit board design, assembly, manufacturing, comments or questions on IPC specifications, or other technical inquiries. IPC also uses TechNet to announce meetings, important technical issues, surveys, etc.

ComplianceNet@ipc.org

ComplianceNet forum covers environmental, safety and related regulations or issues.

DesignerCouncil@ipc.org

Designers Council forum covers information on upcoming IPC Designers Council activities as well as information, comments, and feedback on current designer issues, local chapter meetings, new chapters forming, and job opportunities. In addition, IPC can set up a mailing list for your individual Chapter so that your chapter can share information about upcoming meetings, events and issues related specifically to your chapter.

Gencam@ipc.org

Gencam deals with issues regarding the Gencam™ standards and specifications for Printed Circuit Board Layout and Design.

LeadFree@ipc.org

This forum acts as a peer interaction resource for staying on top of lead elimination activities worldwide and within IPC.

IPC_New_Releases@ipc.org

This is an announcement forum which subscribers can receive notice of new IPC publications, updates and standards.

ADMINISTERING YOUR SUBSCRIPTION STATUS:

All commands (such as subscribe and signoff) must be sent to listserv@ipc.org. Please DO NOT send any command to the mail list address, (i.e.<mail list> @ipc.org), as it would be distributed to all the subscribers.

Example for subscribing:
To: LISTSERV@IPC.ORG
Subject:
Message: subscribe TechNet Joseph H. Smith

Example for signing off:
To: LISTSERV@IPC.ORG
Subject:
Message: signoff DesignerCouncil

Please note you must send messages to the mail list address ONLY from the e-mail address to which you want to apply changes. In other words, if you want to sign off the mail list, you must send the signoff command from the address that you want removed from the mail list. Many participants find it helpful to signoff a list when travelling or on vacation and to resubscribe when back in the office.

How to post to a forum:

To send a message to all the people currently subscribed to the list, just send to <mail list>@ipc.org. Please note, use the mail list address that you want to reach in place of the <mail list> string in the above instructions.

Example:
To: TechNet@IPC.ORG
Subject: <your subject>
Message: <your message>

The associated e-mail message text will be distributed to everyone on the list, including the sender. Further information on how to access previous messages sent to the forums will be provided upon subscribing.

For more information, contact Keach Sasamori
tel 847/790-5315
fax 847/504-2315
e-mail: sasako@ipc.org
www.ipc.org/html/forum.htm

Education and Training

IPC conducts local educational workshops and national conferences to help you better understand conventional and emerging technologies. Members receive discounts on registration fees. Visit www.ipc.org to see what programs are coming to your area.

IPC Certification Programs

IPC provides world-class training and certification programs based on several widely-used IPC standards, including the IPC-A-610, the J-STD-001, and the IPC-A-600. IPC-sponsored certification gives your company a competitive advantage and your workforce valuable recognition.

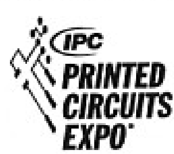
For more information on programs, contact Alexandra Curtis
tel 847/790-5377 fax 847/509-9798
e-mail: curtal@ipc.org www.ipc.org

IPC Video Tapes and CD-ROMs

IPC video tapes and CD-ROMs can increase your industry know-how and on the job effectiveness. Members receive discounts on purchases.

For more information on IPC Video/CD Training, contact Mark Pritchard
tel 505/758-7937 ext. 202 fax 505/758-7938
e-mail: markp@ipcvideo.org www.ipc.org

IPC Printed Circuits Expo®



IPC Printed Circuits Expo is the largest trade exhibition in North America devoted to the PWB manufacturing industry. Over 90 technical presentations make up this superior technical conference. Visit www.ipcprintedcircuitexpo.org for upcoming dates and information.

Exhibitor information:
Contact: Mary MacKinnon
Sales Manager
tel 847/790-5386
e-mail: MaryMackinnon@ipc.org

Alicia Balonek
Exhibits Manager
tel 847/790-5398
e-mail: AliciaBalonek@ipc.org

Registration information:
tel 847/790-5361
fax 847/509-9798
e-mail: registration@ipc.org

APEX® / IPC SMEMA Council Electronics Assembly Process Exhibition & Conference



APEX is the premier technical conference and exhibition dedicated entirely to the electronics assembly industry. Visit www.GoAPEX.org for upcoming dates and more information.

Exhibitor information:
Contact: Mary MacKinnon
tel 847/790-5386
e-mail: MaryMacKinnon@ipc.org

Registration information:
tel 847/790-5360
fax 847/509-9798
e-mail: goapex@ipc.org

How to Get Involved

The first step is to join IPC. An application for membership can be found in the back of this publication. Once you become a member, the opportunities to enhance your competitiveness are vast. Join a technical committee and learn from our industry's best while you help develop the standards for our industry. Participate in market research programs which forecast the future of our industry. Participate in Capitol Hill Day and lobby your Congressmen and Senators for better industry support. Pick from a wide variety of educational opportunities: workshops, tutorials, and conferences. More up-to-date details on IPC opportunities can be found on our web page: www.ipc.org.

For information on how to get involved, contact:
Jeanette Ferdman, Membership Director
tel 847/790-5309 fax 847/509-9798
e-mail: JeanetteFerdman@ipc.org www.ipc.org



Application for Site Membership

Thank you for your decision to join IPC members on the "Intelligent Path to Competitiveness"! IPC Membership is **site specific,** which means that IPC member benefits are available to all individuals employed at the site designated on the other side of this application.

To help IPC serve your member site in the most efficient manner possible, please tell us what your facility does by choosing the most appropriate member category. *(Check one box only.)*

☐ Independent Printed Board Manufacturers

This facility manufactures and sells to other companies, printed wiring boards (PWBs) or other electronic interconnection products on the merchant market. What products do you make for sale?

☐ One-sided and two-sided rigid printed boards
☐ Multilayer printed boards
☐ Flexible printed boards
☐ Other interconnections

Name of Chief Executive Officer/President____________________

☐ Independent Electronic Assembly EMSI Companies

This facility assembles printed wiring boards, on a contract basis, and may offer other electronic interconnection products for sale.

Name of Chief Executive Officer/President____________________

☐ OEM–Manufacturers of any end product using PCB/PCAs or Captive Manufacturers of PCBs/PCAs

This facility purchases, uses and/or manufactures printed wiring boards or other interconnection products for use in a final product, which we manufacture and sell.

What is your company's primary product line? ____________________

☐ Industry Suppliers

This facility supplies raw materials, machinery, equipment or services used in the manufacture or assembly of electronic interconnection products.

What products do you supply?____________________

☐ Government Agencies/Academic Technical Liaisons

We are representatives of a government agency, university, college, technical institute who are directly concerned with design, research, and utilization of electronic interconnection devices. (Must be a non-profit or not-for-profit organization.)



2215 Sanders Road, Northbrook, IL 60062-6135
Tel. 847.509.9700 Fax 847.509.9798
www.ipc.org

ISBN #1-580981-77-1